SYNERGISTIC ENGINEERING ENVIRONMENT
BUILD II
Revision D

USER'S GUIDE

Prepared by
Scott Angster

Analytical Mechanics Associates
Hampton, VA

Revision D
AMA Report No.03-12
NASA Prime Contract No.NAS1-00135
April 2003

Table of Contents

Table of Figures

1 Introduction

1.1 Background

The Synergistic Engineering Environment (SEE) is a graphical environment designed to fuse time dependent data from several different disciplines and analysis codes together, to allow engineers and scientists the ability to more quickly understand a problem trade space. More specifically, the SEE has been designed for the analysis of spacecraft operations. Real time interactive analysis with the models and analysis software allows the users to quickly visualize the results of modifications to various parameters. Build I of the SEE was developed utilizing the International Space Station as the target spacecraft. Capabilities such as rigid body dynamics, robotics, visiting vehicle docking, and field of view obscuration were integrated into one environment to study ISS design and operations.

As the Build I software was further maturing, it became evident that spacecraft other than the ISS could be analyzed within the software. Additionally, spacecraft designs and concepts outside those intended for low earth orbit could be analyzed. As the design of the software continued to change to support the broader range of spacecraft, the Build I architecture could no longer easily support the requirements. This resulted in the redesign of the software, and Build II of the SEE software begun.

Several changes were performed when moving to the second build of the SEE. First, the graphical environment that supported the software had some features that were undesirable. A new API was chosen for Build II. The second change was in the graphical user interface. This software was also changed to provide a better integration method between the interface and the graphical environment, as well as a better distribution method to other systems. The last major change was the move to full cross platform development. The change to the new graphics API and interface API provided a solution to develop for Irix, Windows, and Linux platforms.

In addition to the underlying API support changes, the shift to a generic spacecraft operations and design software tool occurred. A more flexible environment is currently under design and development to handle spacecraft ranging from low earth orbit, to interplanetary vehicles to planetary and moon based spacecraft. The overall architecture for Build II can be found in the SEE Build II Developer's Guide, AMA Report 03-11, April 2003.

1.2 Current Release

The current release of the software is the fourth development release. This document serves as Revision D of the User's Guide for Build II. The current capabilities of the application at this release include solar system visualization, multiple craft loading, integrated ARCD rigid body analysis tools, integrated Comet and Asteroid Protection System analysis tools, collision detection analysis tools, plume cone visualization,

image and move capture, several navigation methods, tethering, craft orbit definition, time control and multiple camera operation.

2 Installation

The Synergistic Engineering Environment is currently developed to support three different platforms. These include Silicon Graphics Irix, Linux (Redhat 8 or equivalent), and Microsoft Win32 platforms (Windows 2000 or Windows XP). The installation procedure used for these platforms various slightly.

2.1 Win32

The Win32 version of the SEE is distributed on a CD and utilizes the InstallShield program to install the software. By inserted the CD into a Win32-based PC the CD should auto start the install script. If the autoplay feature is not enabled, browse the CD and execute the SETUP.EXE file found at the top level. This will start the InstallShield Software, such as that shown in Figure 1. The primary steps for installing are the selection of the root directory for the SEE, the root directory for the user data, and entry of the Gizmo3D license. If a Gizmo3D license is not available at time of installation, a trial license of the Gizmo3D software is installed. This will not limit the capabilities of the SEE software, but a nag screen will be displayed for Gizmo3D periodically until a full license is created.

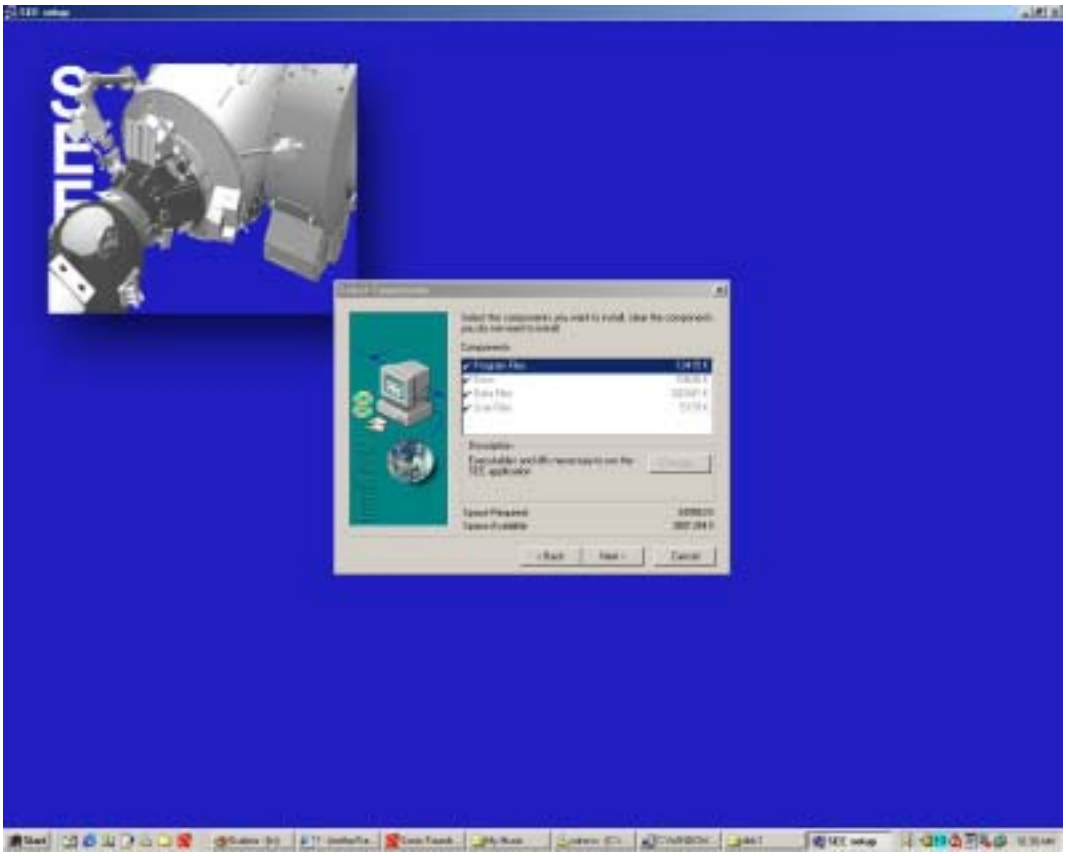

Figure 1. InstallShield Software for the SEE

2.2 Linux and Irix

For the Linux and Irix operating systems, the Synergistic Engineering Environment is currently distributed in three different packets of data. These are the main application packet, the data packet, and a sample user data packet. These are distributed as compressed archived files. Once these files are obtained the user must uncompress and expand files appropriately. The first step is to create a directory for the application that will serve as the base directory for the software. Move all three files into this directory and expand the execution environment and data environment files. The SEE distribution also includes a packet of sample user data, such as sample missions, crafts, and other objects. The user can either expand this file at the same location as the execution and data files or define another location for user specific data.

After the software has been installed into the desired location, a few settings must be made to enable the software to run properly. This involves the setting of two environment variables. The software needs to know where it has been installed onto the local computer system. This is done by setting the SEE_HOME environment variable. Next, the software needs to know where local user files should be stored. This is done by setting the SEE_USER environment variables. These environment variables can be set on the command line or inside a .login or .cshrc file. The example command for setting the environment variables under the c-shell is shown below:

```
setenv SEE_HOME /usr/local/software/seebuild2
setenv SEE_USER /usr/people/seeuser1/data
```

2.3 Directory Structures

When the files are installing or extracting for all platforms, the file structure found in Figure 2 will be located in the SEE base directory. The base level will contain a data directory, a resource directory, a docs directory, and one of three execution directories depending on the platform. If the distribution is a development distribution, a source directory will be present. The data directory contains a data needed for the application such as textures, craft files, craft parts, planet ephemeris, etc. The resource directory contains additional information that could be useful to the user such as additional textures, or model files. The docs directory contains both the User's Guide and the Architecture Guide for reference. The execution directories contain the binary executable files.

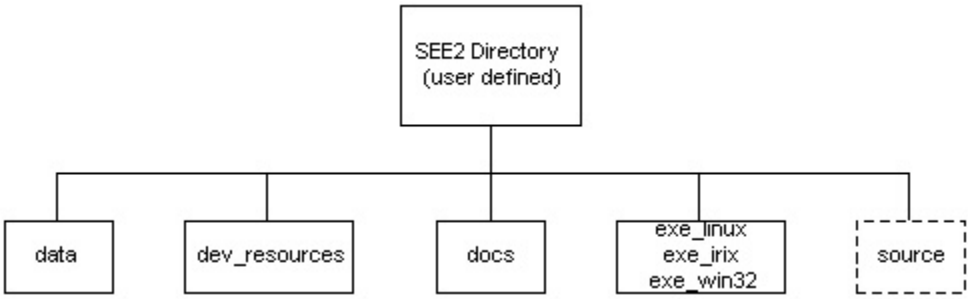

Figure 2. SEE Directory Structure

The layout of the directories for the sample user data can be seen in Figure 3. The mission directory will contain a set of sample missions with varying spacecraft and solar system configurations. Inside the object library directory is a series of directories. These include directories containing sample crafts, parts, planetary systems, simple objects, near earth objects and solar systems.

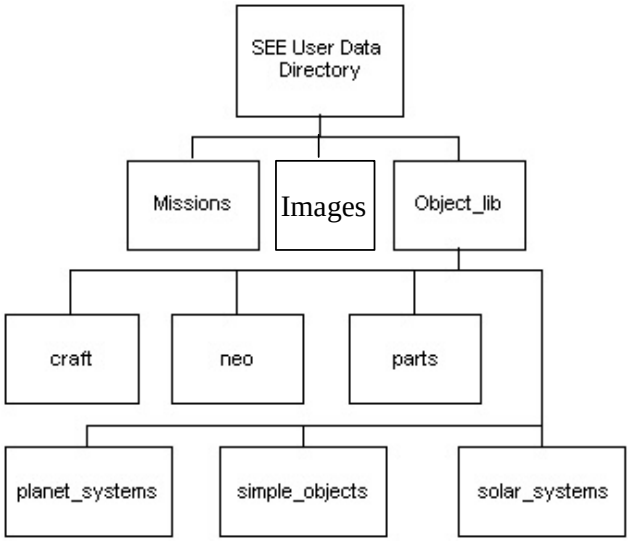

Figure 3. User Data Directory Structure

3 Running the application

3.1 Mission Selection

Upon executing the SEE software from the appropriate execution directory, the SEE software will bring up a Mission Wizard window as seen in Figure 4. The user is presented with three choices: New Mission, Resume Mission, and Load Mission. New mission allows the user to create a mission, defining the solar system and adding predefined crafts. Resume mission loads in the last mission the user worked on, including any unsaved changes upon exiting. Load mission allows the user to browse the mission directory to load in a mission. The last mission loaded is the default mission in the interface. This will be the last saved version of the mission.

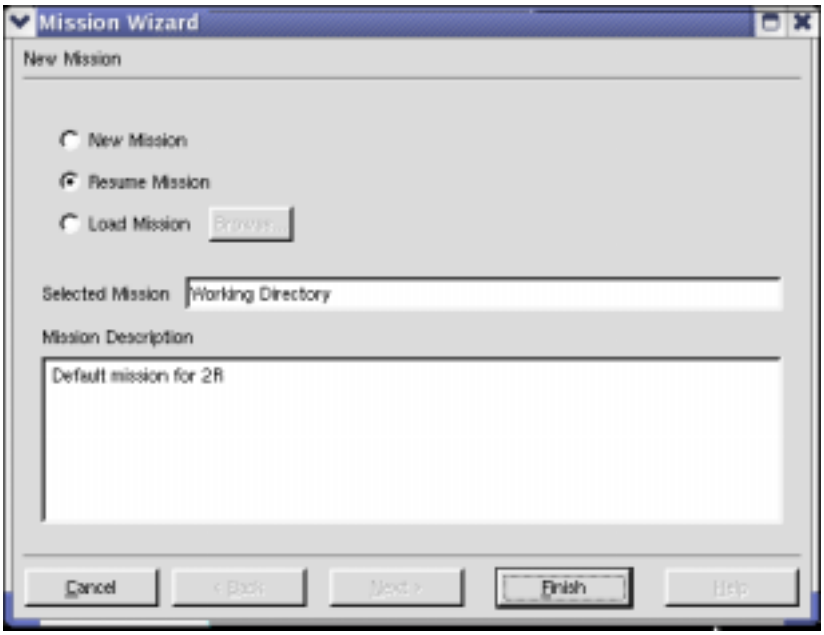

Figure 4. Mission Wizard

3.1.1 New Missions

A new Mission can be created by selecting the New Mission option from the Mission Wizard and pressing Next. The user will be prompted to enter in the basic properties for the new mission. These include a title for the mission, a description of the mission, and the starting Epoch date. This window can be seen in Figure 5. Next, the user must setup the solar system configuration. Currently, the only option is to create a new solar system configuration as seen in Figure 6. In future releases, the user will be able to save a given configuration for future incorporation into a mission. To create the mission, the user selects which planets and moons should exist for the mission. The first window, shown in Figure 7, allows the user to select from a stock list of planets and moons that are provided by the SEE. Once these are selected, the user has the choice of importing a custom planet or moon. The file format and details of how to create a custom planet can be found in the Developer's Guide. The last requirement of the user is the addition of crafts to the scene, as seen in Figure 8. By selecting the Add button, the user can browse their directories and select a craft directory that defines the actions and geometry of a craft. The details of setting up a craft specification can be found in the Developer's Guide. Once the solar system and the various crafts have been setup, the user can select Finish to load in the data and visualize the mission.

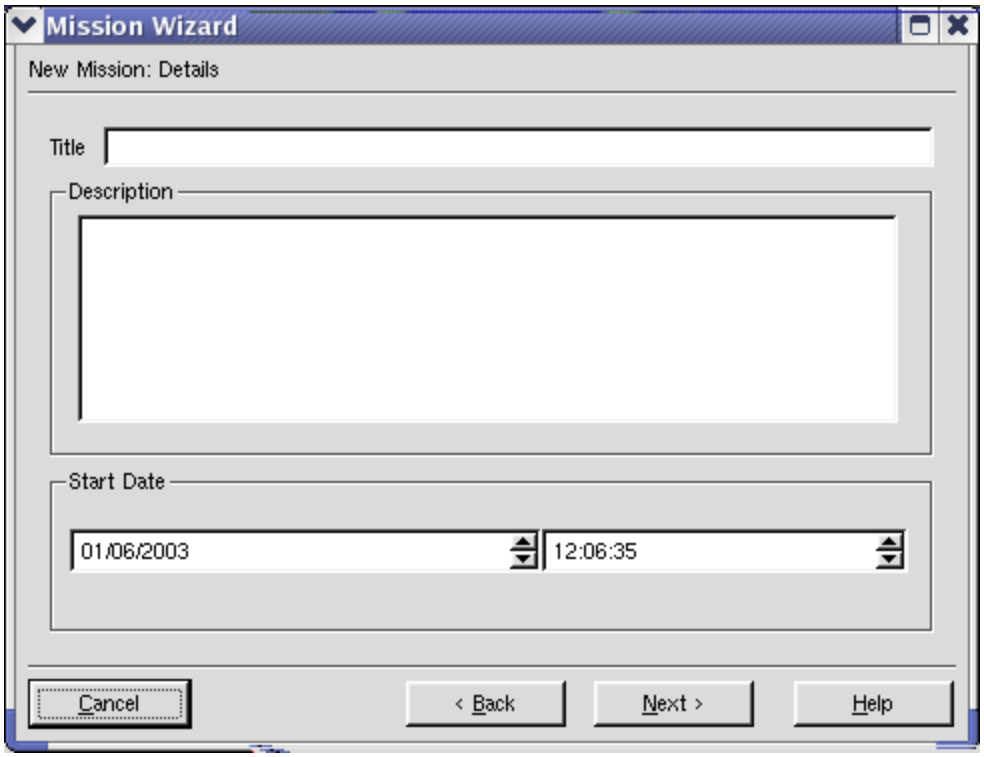

Figure 5. New Mission Title Window

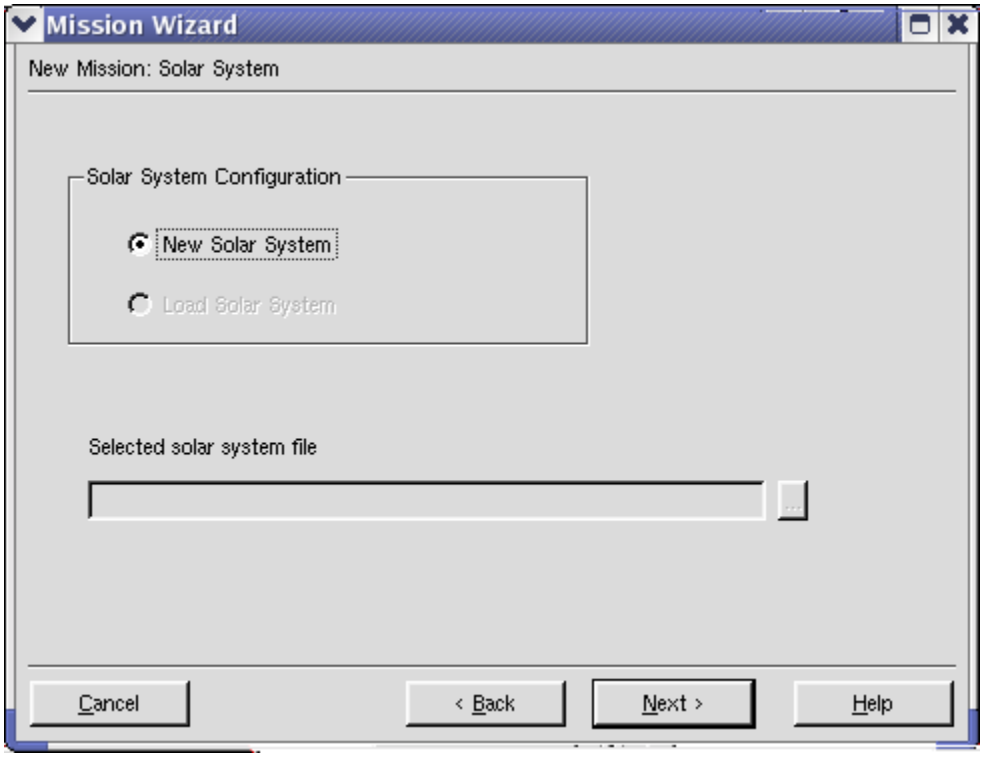

Figure 6. New Mission Solar System Window

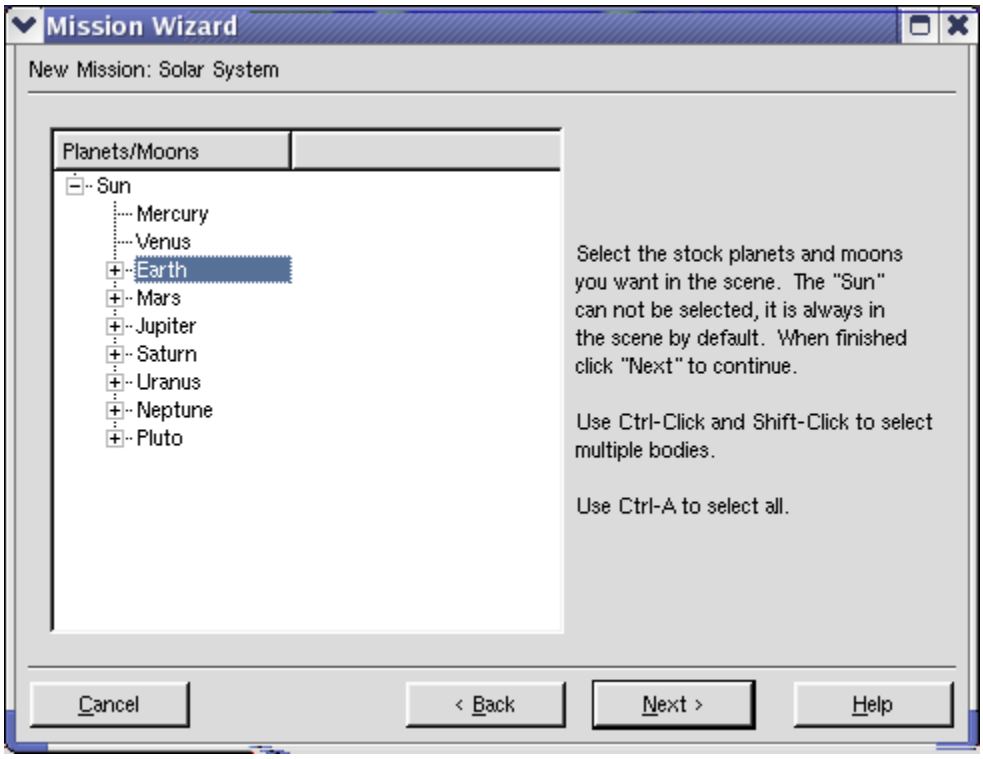

Figure 7. New Mission Wizard Selecting Planets

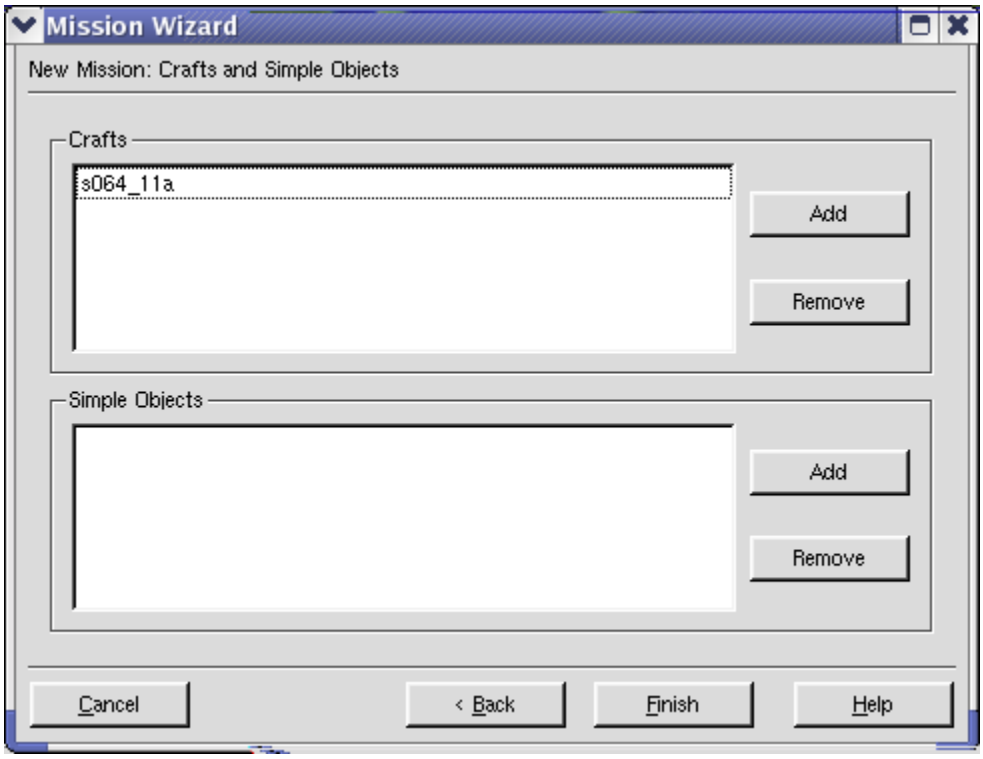

Figure 8. New Mission Wizard Adding Crafts

3.1.2 New Craft

In the creation of a new mission, the SEE currently does not provide a mechanism to for creating a new craft at the same time. Only existing crafts can be added to the scene. However, once a mission is created and is active, the user can create a craft and add it to the current mission by selecting the New option under the Mission menu as shown in Figure 9. By selecting New, the menu will expand such that the user can then select Craft. This will provide the user with a New Craft Wizard, as shown in Figure 10.

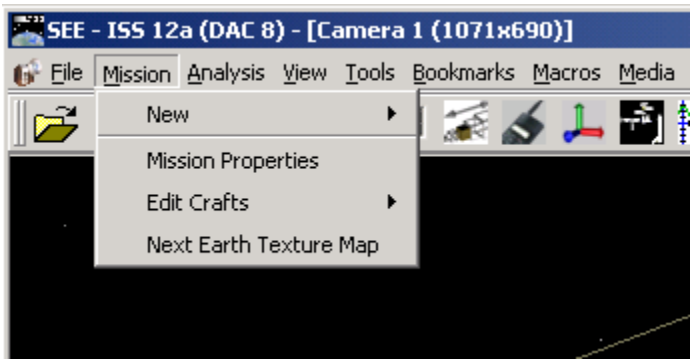

Figure 9. Mission Menu

This wizard provides a method for the user to add crafts to the scene without the need to edit the underlying craft files. To allow for this ease of use, the wizard provides functionality for setting only the most frequently used options for a craft. These options will provide all the needed functionality for the majority of the users. To setup the advanced capabilities the users can use the edit craft option once the craft is created. This option is also found under the Mission Menu.

The settable parameters for setting up a new craft are listed in sequential order in the pages of the wizard. To quickly setup a craft, at any time, the user can repeatedly page through the wizard by hitting the Next button and then the Finish button on the last page. If a craft is created in this manner it will be assigned default values for the necessary attributes and placed in the scene.

On the first page of the wizard the details for the craft are set including the name, primary, parent reference frame, and any documentation that should be included with the craft. On the second page of the wizard, as seen in Figure 11, the user sets up the dynamics model and epoch date for the new craft. The last step allows the user to set the graphical properties of the craft including selecting a DDEX file to setup the model of the craft if that is appropriate. This can be seen in Figure 12. Several steps have been taken to insure that the names for new crafts are unique such as appending numbers to repeated names. This allows for proper storage of the new craft when

missions are reloaded. Once the craft is added to the scene, it will be available in all tether, scale, and edit menus.

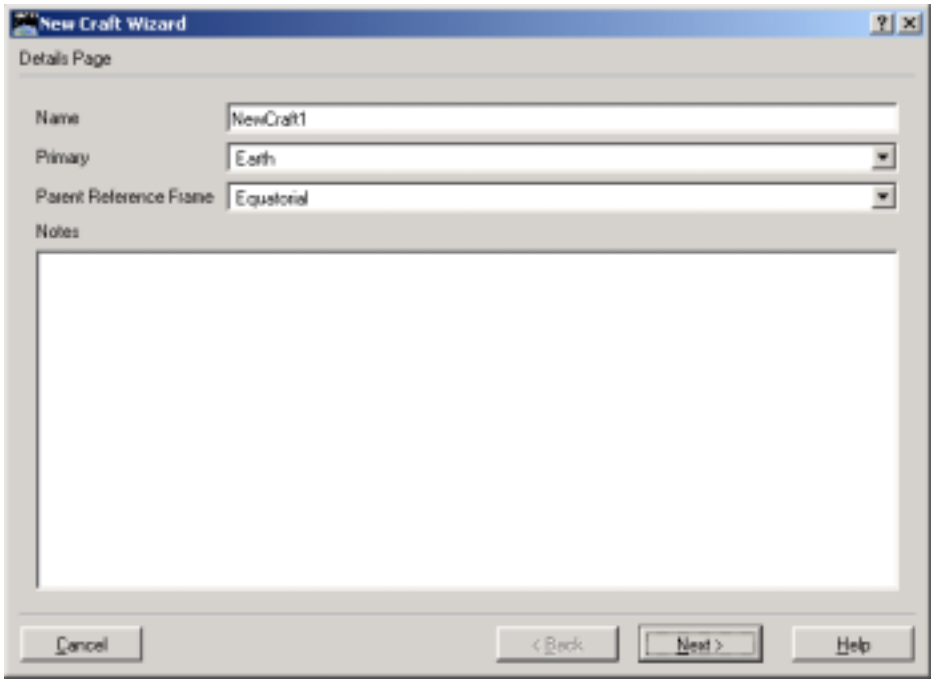

Figure 10. New Craft Wizard

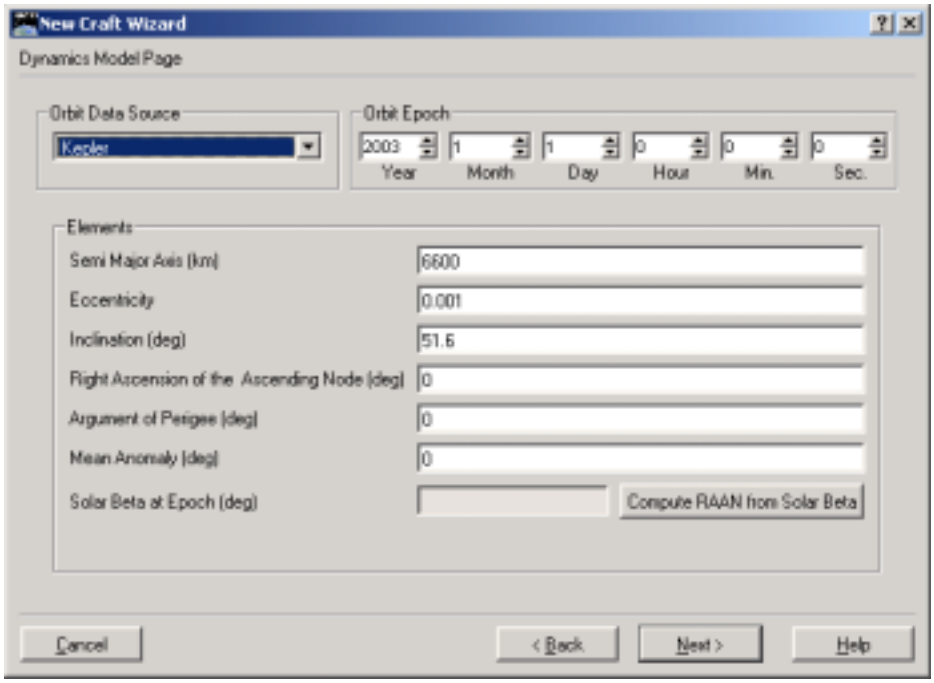

Figure 11. New Craft Dynamics Properties

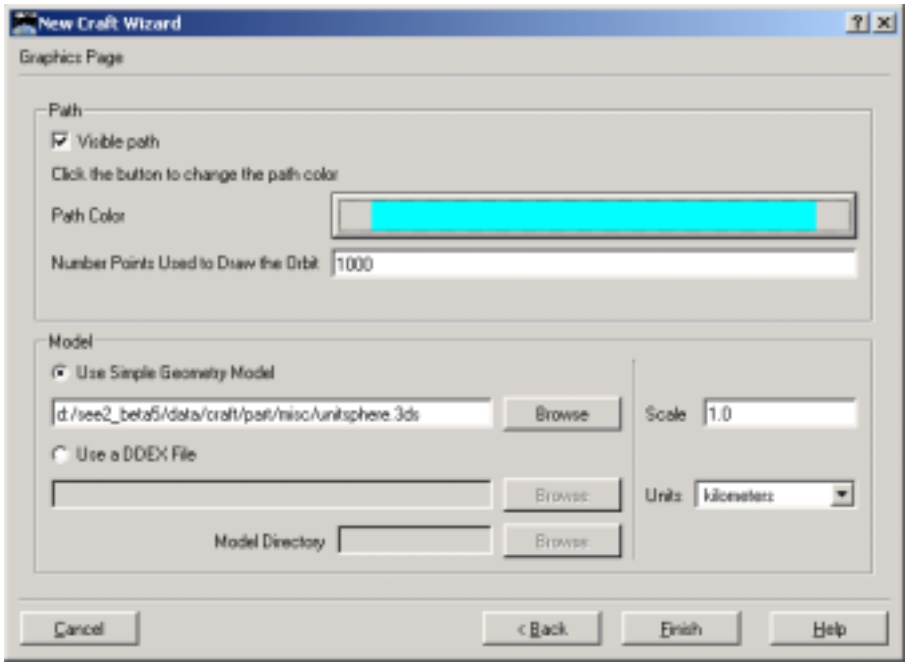

Figure 12. New Craft Graphics Properties

3.1.3 Loading

Any saved mission can be loaded at the startup of the application, or by selecting load mission from the File pull-down menu, as seen in Figure 13. The user will be presented the Mission Wizard as discussed in 3.1.

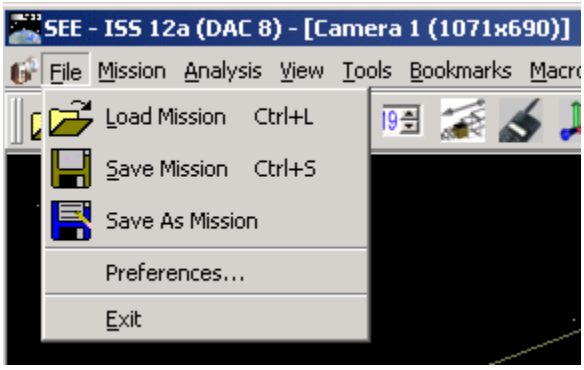

Figure 13. File Menu

3.1.4 Saving

The user can save a mission by selecting the Save Mission option from the File Menu, as seen in Figure 13. Currently this option saves the mission into the directory from which it was loaded. Future releases will allow the user to provide a new mission name.

3.2 Interface Layout

The SEE environment is primarily split into five sections. These five sections can be seen in Figure 14. The Main Menu provides access into some of the main graphical interfaces of the application. These include cameras, windows, craft manipulation and system information. Under the Main Menu is the Toolbar. This Toolbar contains quick access icons to some of the more commonly used features such as new cameras and navigation. At the right hand side of the screen is the vertical graphical user interface. It is split into two sections. The upper section contains a slider that can control one of several features. These include headlight and field of view. The active item is selected by using the pull down menu under the slider. The lower section of the vertical interface contains a scroll wheel that can also control several capabilities such as scale and time This is currently unavailable. Again these can be selected using the pull down menu. The lower horizontal interface contains the main user interfaces of the application. These include the navigation menu, the tether menu and the Time Menu. Each of these will be discussed in detail later in the documentation. At the center of the screen is the graphical window. This window represents a camera view into the scene. Multiple cameras will create multiple windows.

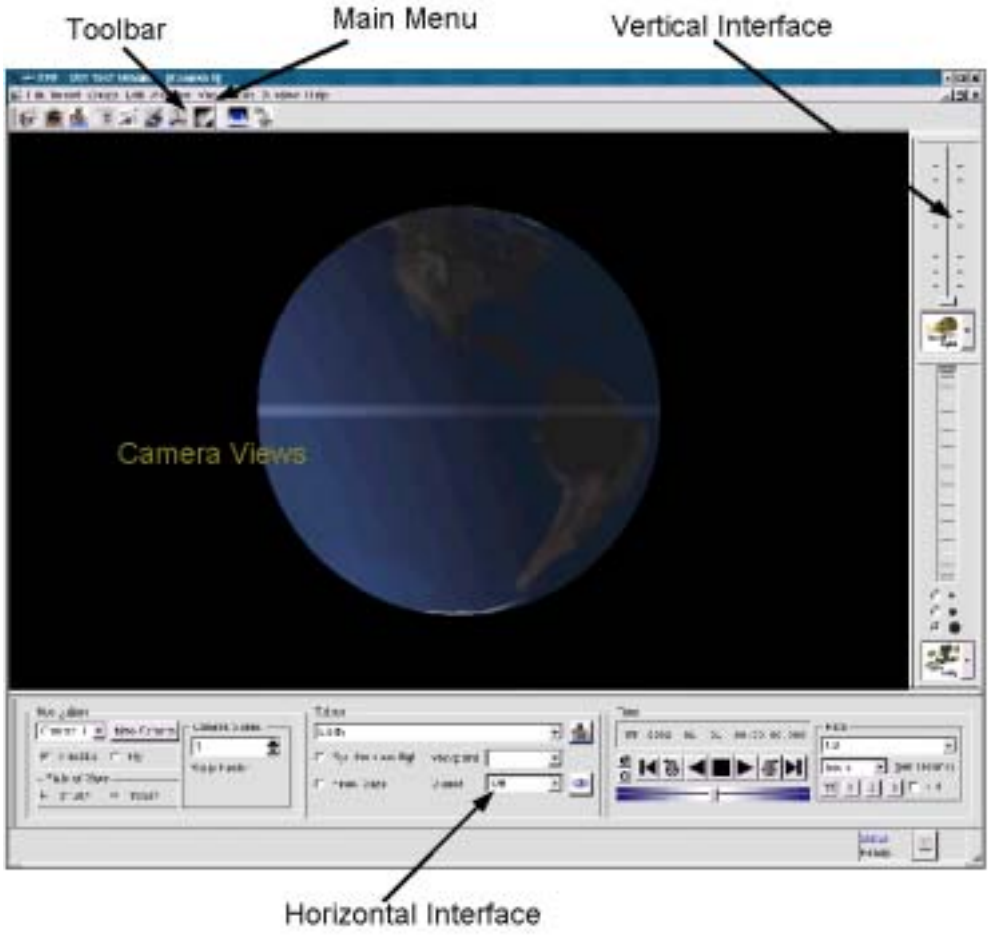

Figure 14. SEE Environment Layout

The Main Menu of the interface is fixed to the main window of the application. The toolbar, the vertical interface and horizontal interface can be removed from the window by selecting and dragging the selected interface at the double line at either the top or left side of the interface. This allows more room at the center of the screen for the camera view widows. They can be placed back into the window by dragging them back to their attach points in the main window and releasing the mouse button.

4 Basic Operations

4.1 Cameras

A camera represents a view into the overall scene contained within the current state of the application. By default an initial camera is created at the startup of the application. The user can create an additional camera by selecting either the New Camera button located on the horizontal menu, selecting New Camera from the Window pulldown or by selecting the camera icon on the toolbar. These can be seen Figure 15 and Figure 16. Multiple windows can be organized either by direct manipulation by the user or by selecting Cascade or Tile under the Window pulldown from the Main Menu. A sample of two windows can be seen in Figure 17. The user can interact with each window individually, allowing different views into the scene. Time, however, will be constant throughout all windows. The active window is highlighted and is listed in the horizontal menu under the Navigation section. It should be noted that additional cameras could impact the overall performance of the application.

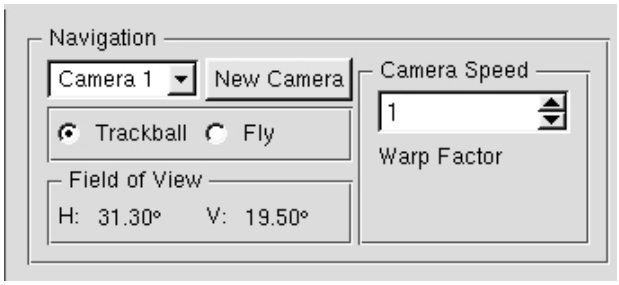

Figure 15. Navigation Menu

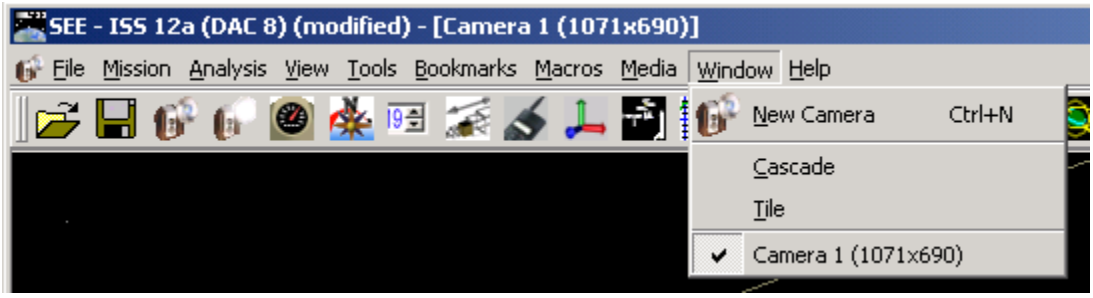

Figure 16. Window Menu

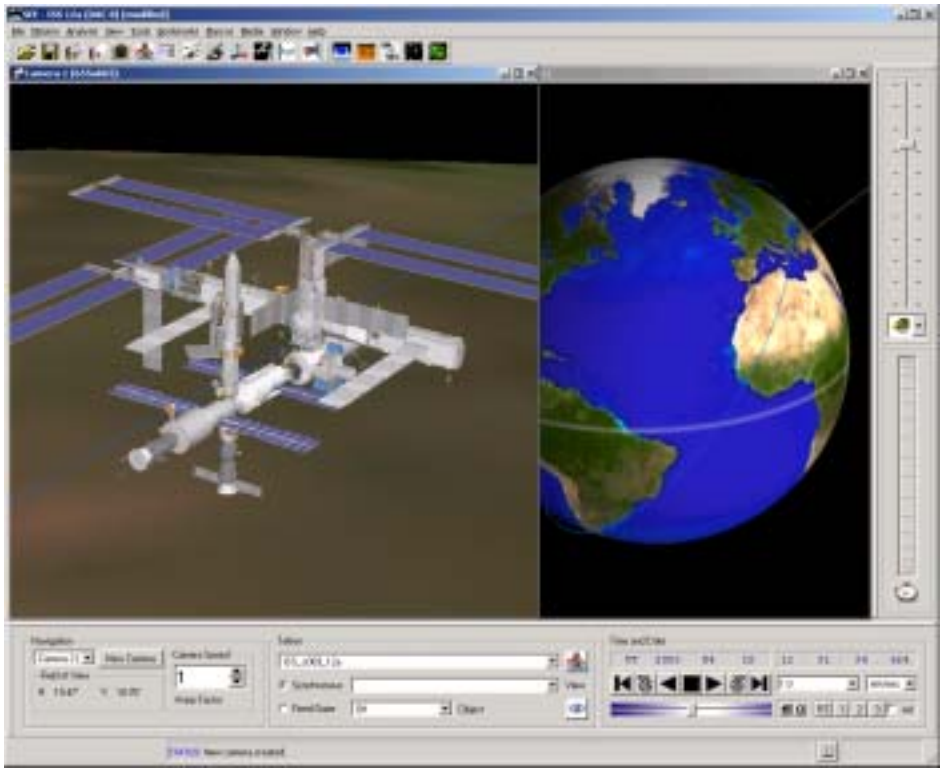

Figure 17. Multiple Camera Views

4.2 Tethering

One of the primary methods of navigating the scene is the use of tethering. Tethering allows the user to quickly move a camera relative to a particular object in the scene. To enable the ability to select an object to tether to, the user must select either the Navigation button on the horizontal menu or the Navigation icon on the toolbar. The button will bring up the navigation window with the tether tab active, as seen in Figure 18. The toolbar will bring up the navigation window with the mode tab active and the user must select the tether tab. Once open, the navigation/tether window will present the user with a hierarchical based menu of selectable items. By expanding the various items, the user can select an object to tether to. For the object selected, a list of available reference frames will be presented. The user can change the reference frame by selecting one from the pulldown menu. For each reference frame the user can also choose to display a set of axes for the reference frame by selecting the "Show axes" option. An additional option for tethering that can be set prior to pressing select is the use of synchronous flight. This will lock the user to the rotation of the object. The user can select the "Synchronous flight" option and select which axes to view the object from the pulldown menu. This option is also available from the horizontal menu after tethering. Once the user presses select, the camera will move relative to that object and place the object name identifier into the horizontal menu under Tether. As long as the user stays tethered, all navigation will remain

relative to that object, even if time is running. During tethered navigation, the motion of the camera is in the trackball mode. To move to a specific viewpoint of an object, a pull down list is available with several predefined axes of gaze, as seen in Figure 19. This that will automatically move the camera to that axis relative to the object and point the camera at the object. If the user wishes to stay tethered to an object but fix the gaze of the camera towards a different object, the user can select the gaze option, as seen in Figure 19, and select an object from the object pulldown.

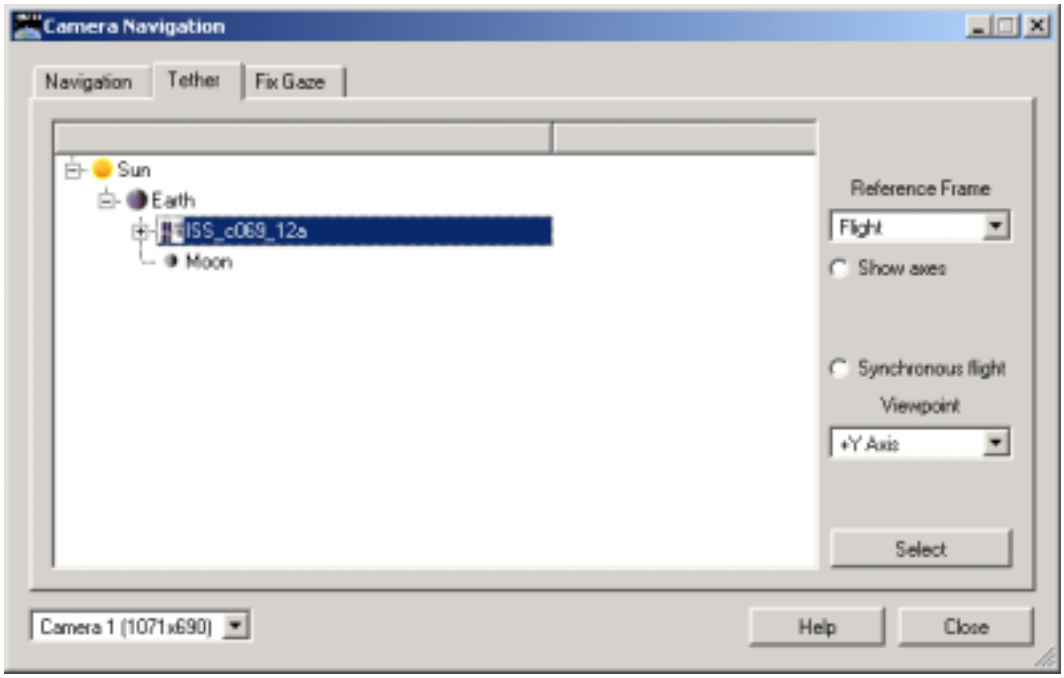

Figure 18. Tether Navigation Window

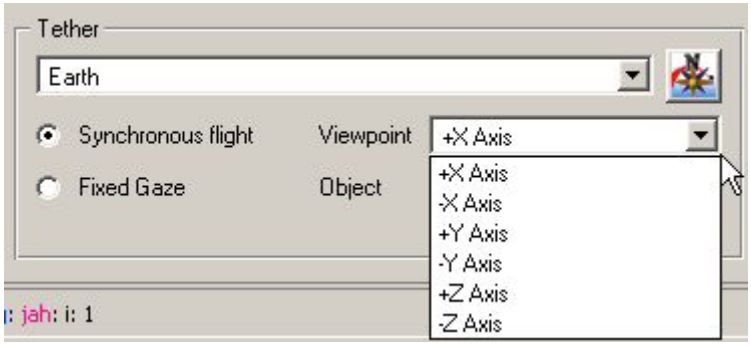

Figure 19. Tether Viewpoints

Once an object has been selected for tether, an additional access method to tethering is available. The horizontal tether menu will store each item selected into a pull down

selectable menu, as seen in Figure 20. Each item can be picked, including Off, to provide easy access to repetitive tether selections.

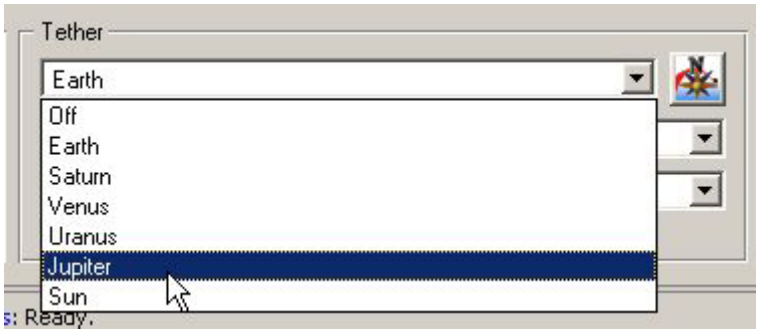

Figure 20. Stored Tether Items

4.3 Navigation

Each camera can be manipulated by the user to navigate through the scene. Currently the only mode of navigation is the Trackball mode. In Trackball mode, the motion is constrained to move around a center point, as if the camera is connected to this center point via a string. By default the camera is pointed at the center during this motion. This point of rotation is based on the currently selected object to which the camera is tethered. By pressing the left mouse button and moving the mouse left and right, the scene will rotate left and right. Likewise, by moving the mouse up and down, the scene will rotate up and down. By pressing the right mouse button and moving the mouse up or down, the camera can be moved in or our relative the rotation point. The middle mouse button can be used to roll the camera by holding the button down and moving the mouse left to right at the top and bottom of the scene or up and down on the left and right portions of the scene. The speed of these motions is set by the Camera Speed setting as was seen in Figure 15.

These motions are also mapped to the keyboard. An upward and downward rotation of the scene can be achieved by using the up and down arrow keys. Left and right rotations are achieved with the left and right arrow keys. Zooming in and out can be controlled using the "w" and "s" keys. Roll is also available using the "q" and "e" keys.

4.4 Time

4.4.1 Running Time

Time within the environment is controlled through the Time Interface on the horizontal scroll bar, as seen in Figure 21. Located at the top of the menu is the current time being represented within the application, starting with the year, then the month and day and ending with the time of day, down to the millisecond.

Time is controlled through the selection of the various VCR style buttons under the current time display. To start time running, the left and right arrows change the time at a given rate either forwards or backwards. Time is stopped by selecting the square. To momentarily run time either forwards or backwards, the user can press and hold the buttons with the reel icon. When released, time stops. Additional time control is available through the right GUI when the scroll wheel is set to the SimTime setting.

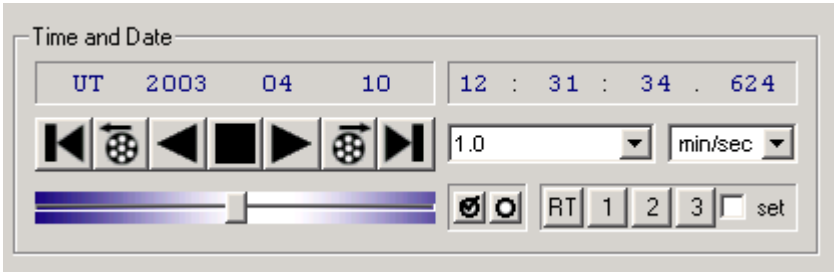

Figure 21. Time Interface

4.4.2 Time Markers

At any given time, the user can create a marker in time by pressing the set marker button. To return to that time, the user can press the buttons on either end of the VCR button set with the arrow and straight line. These buttons immediately move time to the next or last time marker that has been set relative to the current time. Once at that time, the marker can be removed by selecting the delete marker button. If the delete button is selected, a confirmation window, as seen in Figure 22, will present the closest marker set to confirm the right marker is being deleted.

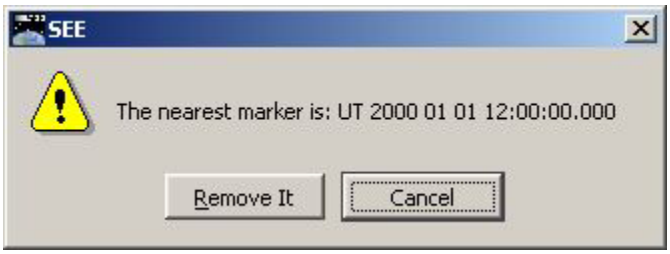

Figure 22. Delete Marker Confirmation

4.4.3 Time Rates

The current rate of time is shown under the Rate interface, as seen in Figure 21. To change the rate of time, the user can either utilize the dynamic scroll bar under the VCR buttons or use the predefined rate selection method. When dragged left or right the scroll bar will decrease or increase the rate of time. The further the scroll bar is dragged, the faster the change in the rate of time. When released, the

scroll bar automatically returns to the center. The current rate will be updated in the Rate interface. The rate can also be changed by using the pull down menu and selecting one of the predefined numbers, such as 1 or 10, as seen in Figure 23. Then the units associated with this rate can be chosen using the pull down menu for the units, as seen in Figure 24. This combination represents the time per second rate within the application when time is running.

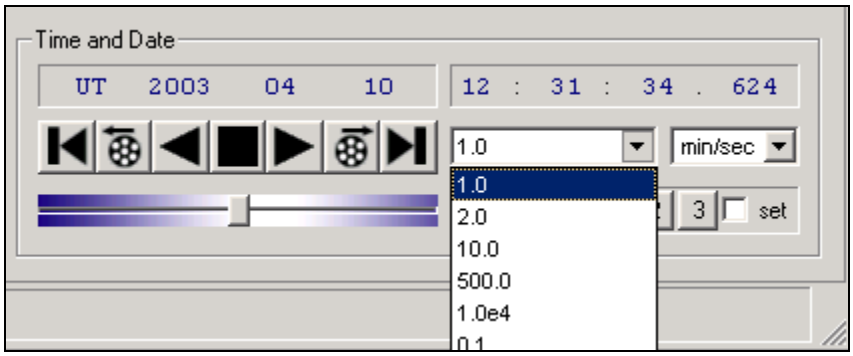

Figure 23. Time Rate Selection

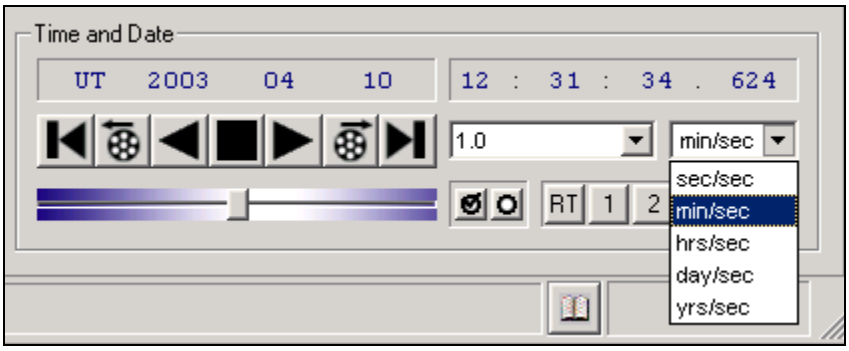

Figure 24. Time Rate Units

Rates can be saved into markers by selecting the Save checkbox and pressing one of the three marker buttons. To return to one of the rate setting, simply press the rate marker desired. The RT button returns the application to real time, or one second per second.

4.4.4 Setting a Time

In addition to running time forwards and backwards at given rates, the user can directly set the application time. This is performed through the SimDate Controller, as seen in Figure 25. This controller is activated by selecting SimDate Controls under the View menu, as seen in Figure 26. A set if seven scroll windows will be presented. Each individual value can be set. The application will automatically change the time based on the inputs given in the Control window. If the Control window is left open during the running of time, the

application time will be updated in the window, but some loss of performance may occur.

Figure 25. SimDate Controller

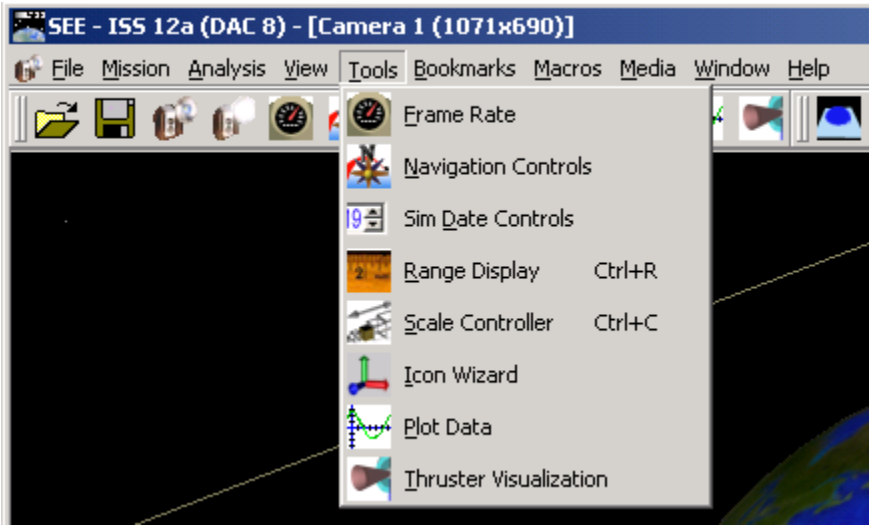

Figure 26. Tools Menu

4.5 Scaling

Due to the often large scale of an environment being analyzed within the SEE, it may be necessary to scale up objects within the SEE to be able to see them. This capability is provided to the use through a scaling interface. This interface is started by selecting the Scale Controller from the Tools pulldown on the Main Menu, as seen in Figure 26 or from the Toolbar.

Selecting Scale Controller will bring up the Scale Control Window shown in Figure 27. The top section of the window displays all of the scaleable items within the scene in a hierarchical list. This includes all planets, moons, crafts, craft orbits, and icons. Next to each item is the current scale of each item and its current size. The size represents the radius of a bounding sphere of the object. The user can select an item and modify its value by either typing a value into the window or by using the scroll knob on the right hand side. A value for either a percentage scaling or a direct value can be entered. The speed of the scroll bar can be set by selecting small, medium, or large increments by checking the appropriate radio button. Bookmarks of the different scale setups can be set and saved using the create and delete buttons. A unit scale bookmark is provided to the user. A sample scaling of the International Space Station around the Earth can be seen in Figure 28.

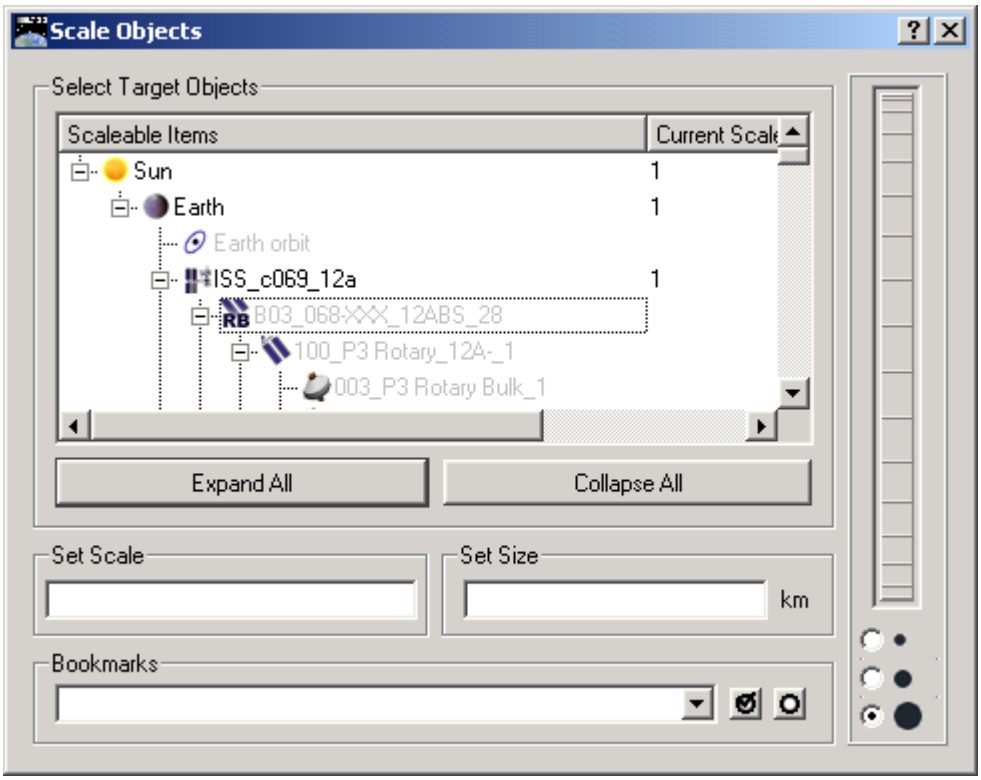

Figure 27. Scale Control Window

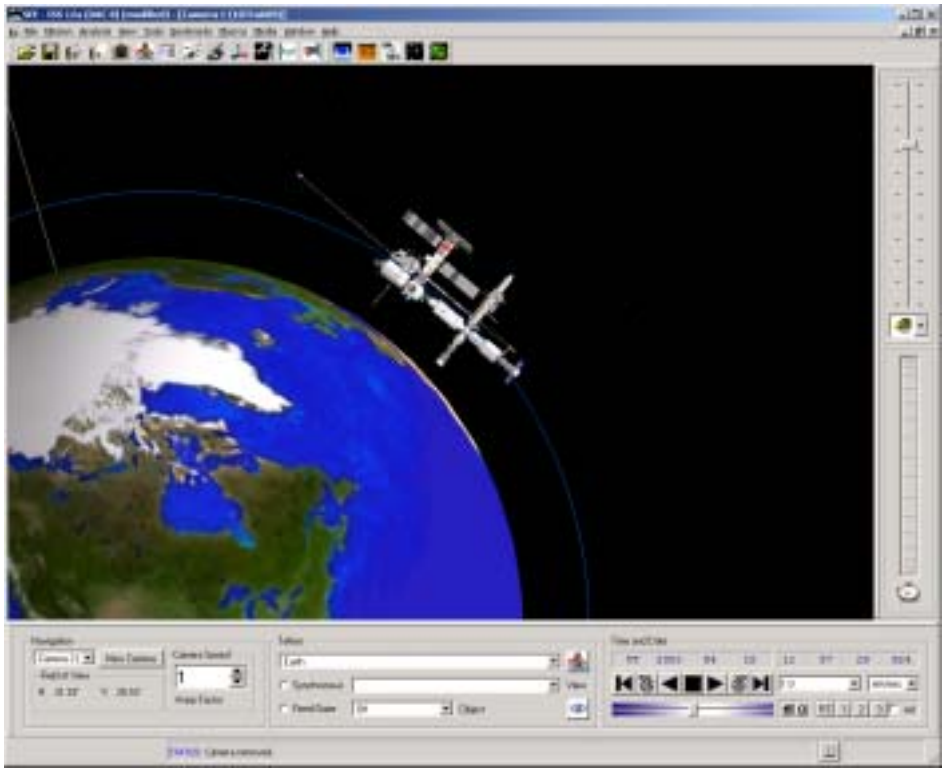

Figure 28. Sample Craft Scaled

4.6 Icons

It is often necessary to visualize the numerous references frames and their position and orientation relative to one another to gain a better understanding of a given situation. Within the SEE, the user can create various icons including coordinate system axis and arrows. By selecting the Icon Wizard from the Tools pulldown on the Main Menu as seen in Figure 29 or from the toolbar, the Icon Wizard is displayed. This can be seen in Figure 30. The first step in adding an icon is the selection of type. Either an XYZ coordinate system or an arrow can be added. If the user selects the XYZ axes, the window shown in Figure 31 will be displayed. This presents a list of all available objects to which a reference frame can be drawn. Upon selection of an object, a list of references associated with that object is provided. The user can also select the style of reference frame so that those in close proximity to one another can be identified. Once finish is chosen, the arrows will be drawn at the specified location. If the arrows are not scaled to the proper size, they can be scaled within the scaling interface described in Section 4.5.

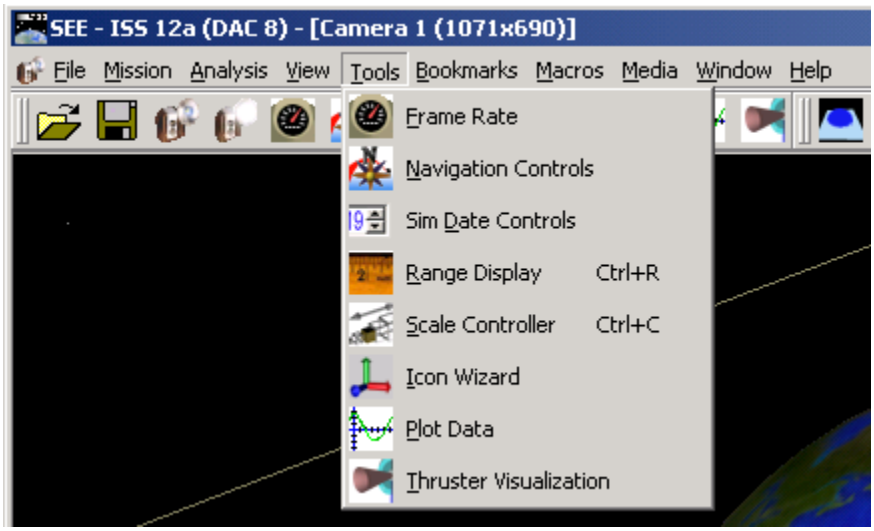

Figure 29. Tools Menu

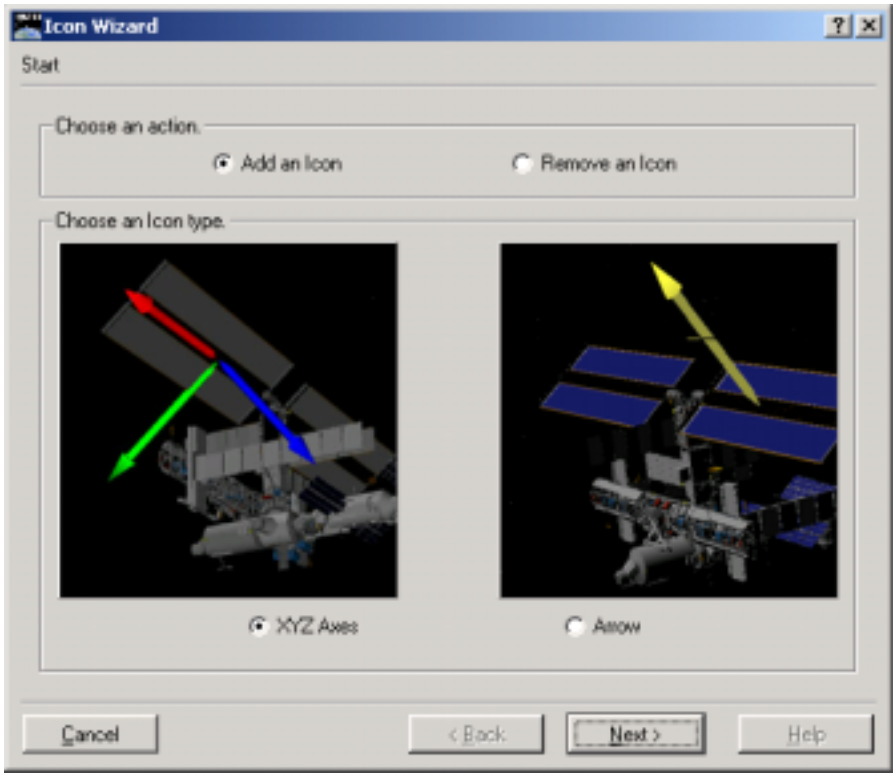

Figure 30. Icon Wizard

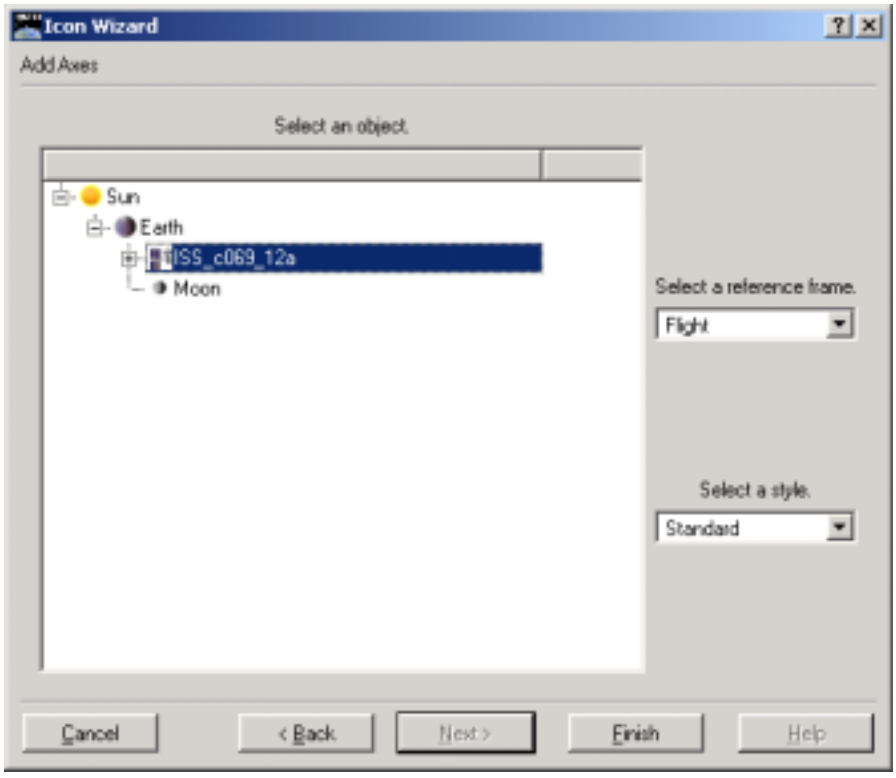

Figure 31. Adding Axes

If the user selects the arrow option, pressing next will present the window shown in Figure 32. In this window, the user must select the type, color, and style for the arrow. Pressing next will bring up the window shown in Figure 33. This window presents all of the objects in the scene that the user can add an arrow to. On the left hand side, the user must select which object the arrow should be attached to. Then the user must select from the right hand side the object to point at if it is a "Point at Object" arrow being added. As with icons, arrows can be scaled using the Scale Controller.

To remove an icon, the user must launch the icon wizard again. By selecting the type of icon to remove and pressing next, a list of all currently created icons of that type is presented. Once the user selects the appropriate icon and presses finish, the icon is removed.

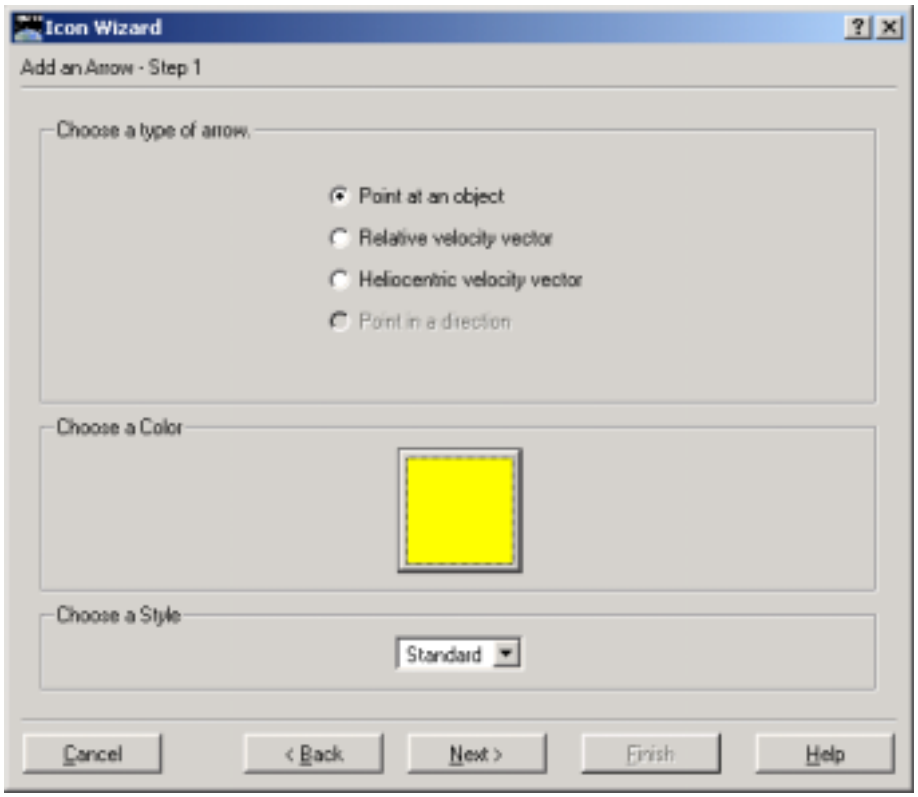

Figure 32. Adding Arrows

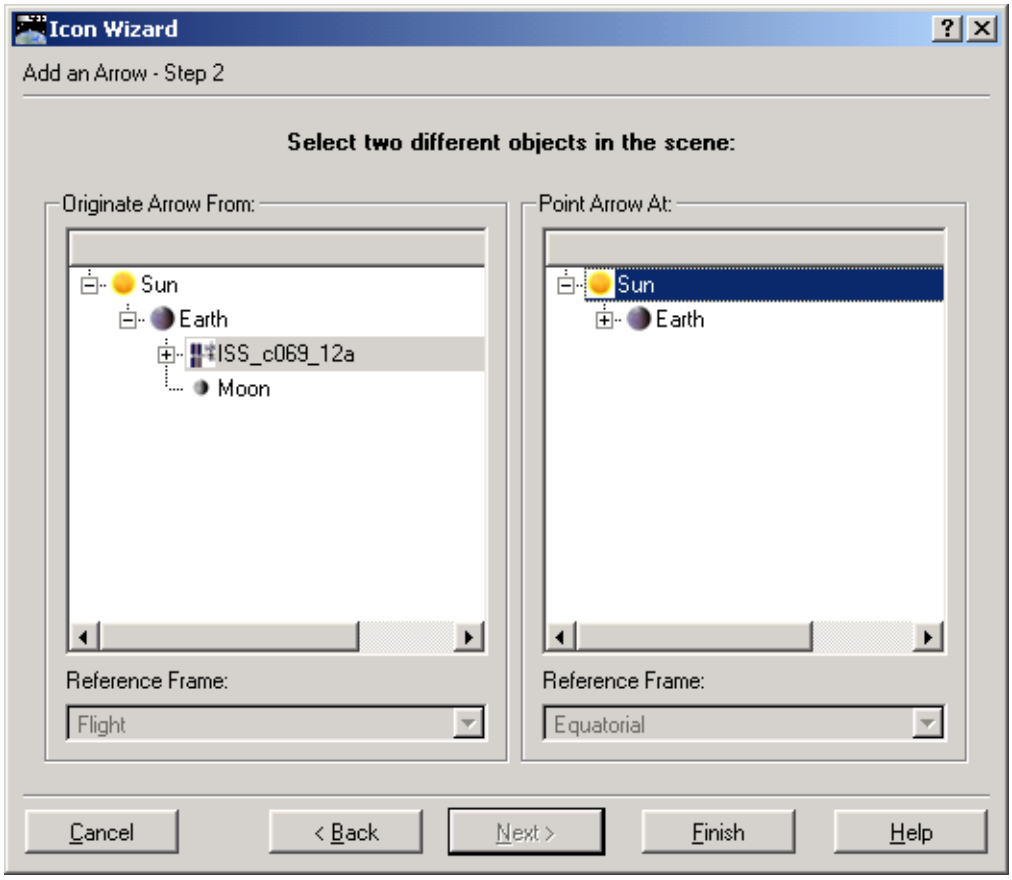

Figure 33. Selecting Object for Arrows

4.7 Bookmarks

Within the SEE, bookmarks allow the user to store and retrieve the current settings of the application. Currently four different settings can be stored. These are time, position, icons, and scaling. Each of these settings can be saved independently or grouped together in a bookmark. The bookmark menu is located on the main horizontal menu as seen in Figure 34. By pressing the Add Bookmark option, the window shown in Figure 35 allows the user to name the bookmark and select which of the four parameters should be recalled when this bookmark is selected. The chosen settings are used as the default the next time Add Bookmark is chosen.

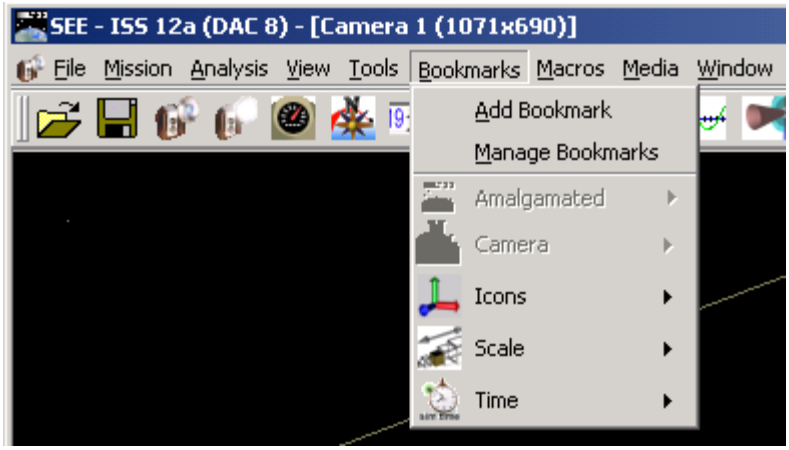

Figure 34. Bookmarks Menu

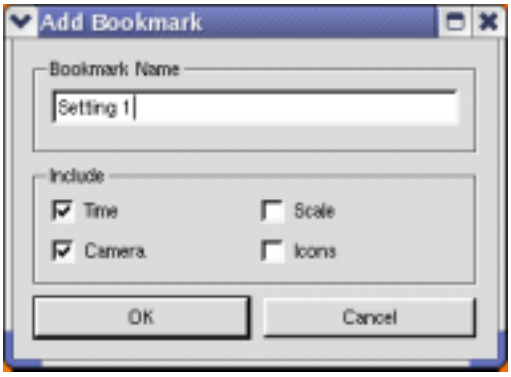

Figure 35. Adding a Bookmark

4.8 Plotting

An internal capability to plot data is included within the SEE. The current plotting capability allows the user to plot Euler angle sequences between two reference frames, the angle between any two vectors, and the plume firing data if plume data is present.

To create a plot for the first two options, the first step is to add any vectors or reference frame icons that would be required for such calculation to the scene. Next, the user starts the Plotting Wizard by selecting the plotting tool from the tools menu or from the toolbar as shown in Figure 36. The main plotting window, as shown in Figure 37, will be displayed where the user selects what type of plot to be created. Next the user must select either a set of vectors or reference frames depending on what type of plot was selected. This is shown in Figure 38. Only those objects in the scene which have icons associated with it are displayed. The user must then specify the time period for the length of the plot. Additionally, the user can specify the name to be drawn on the plot. This window is shown in Figure 39. Once the user specifies the needed information, the SEE will create the plot by taking control of time and

creating the required data. The creation of the plot can be speed up by minimizing the camera window while the data is being generated. A resultant plot is shown in Figure 40. At the bottom of the plot, the user has the option of printing or saving the plot. Plots are saved as standard image file formats such as PNG and JPG.

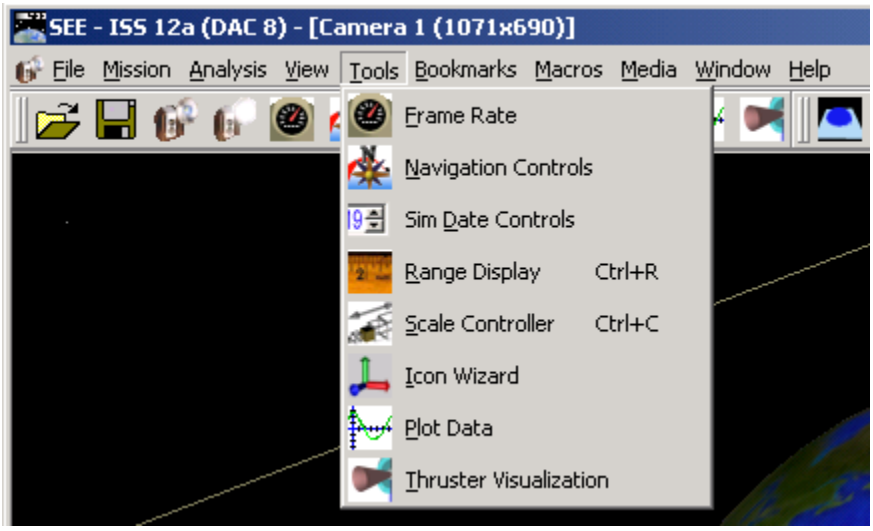

Figure 36. Tools Menu

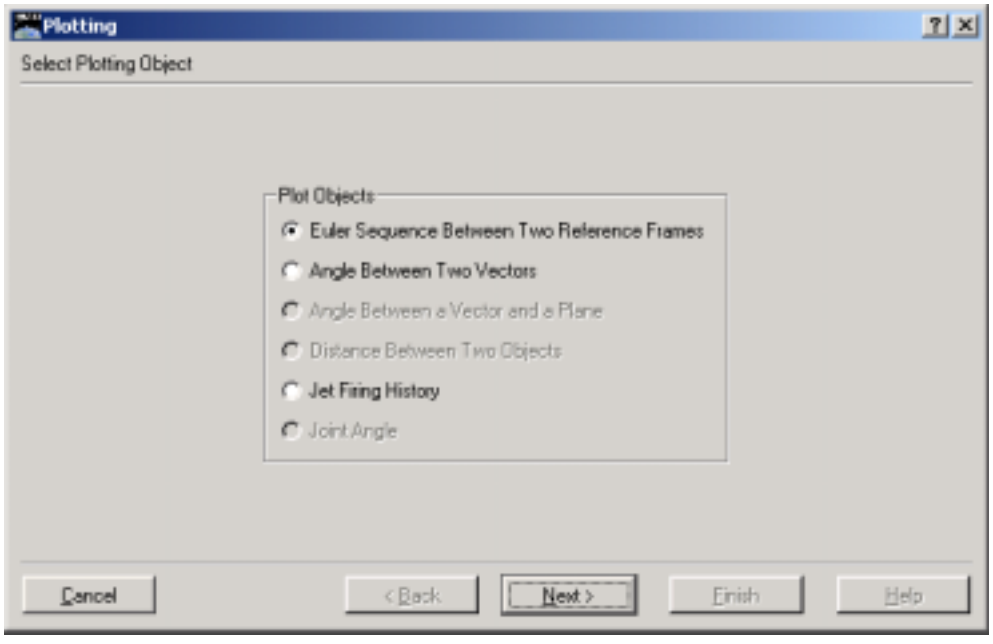

Figure 37. Main Plot Window

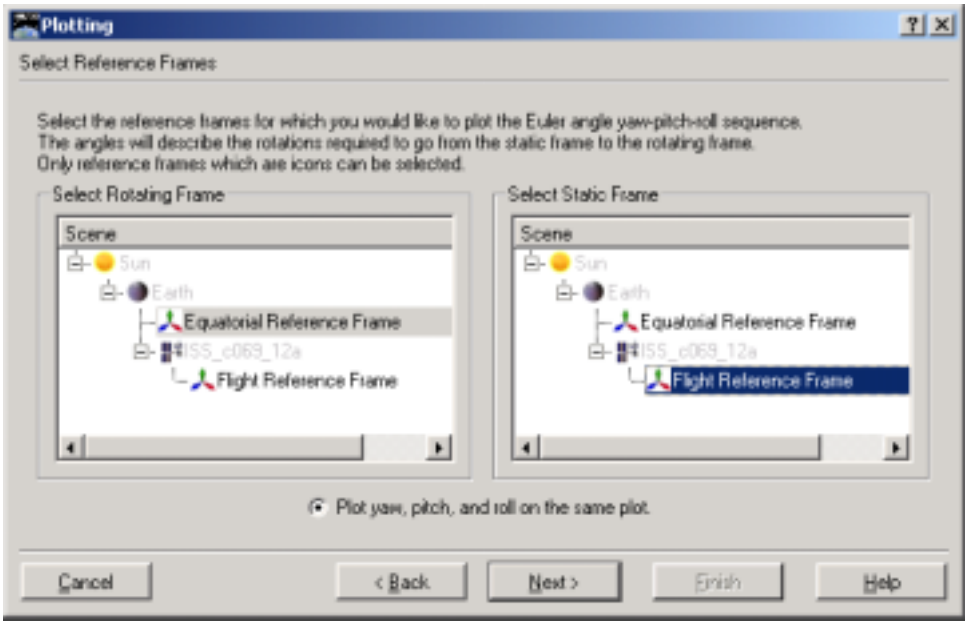

Figure 38. Selecting Plot Reference Frames

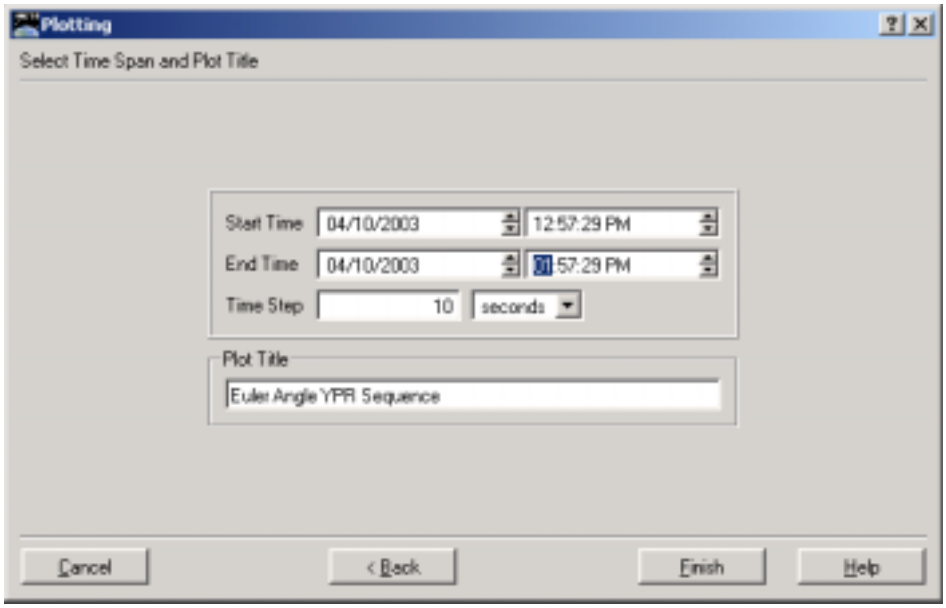

Figure 39. Plot Time Parameters

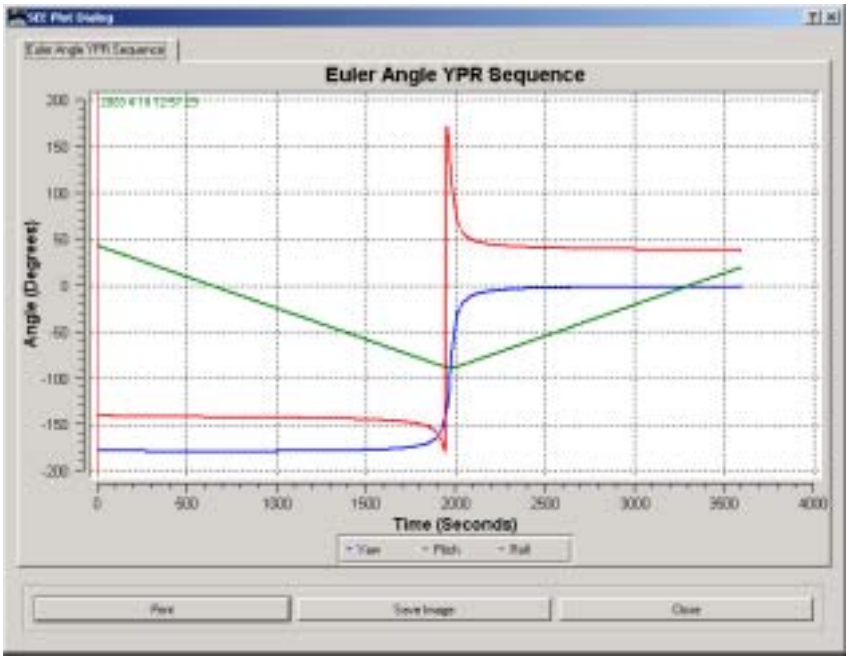

Figure 40. Euler Angle Plot Generated from SEE

If the current mission has jet firing history loaded, the user can plot the jet firings as a function of time. By selecting this option from the main plotting window, the user is given the choice of which jets should be included in the plot. This can be seen in Figure 41. Once these are selected, the user can select to plot all the jet firings that are loaded for the jets or limit the time of interest. The user can also specify a new title for the plot. This window and a resultant sample output can be seen in Figure 42 and Figure 43.

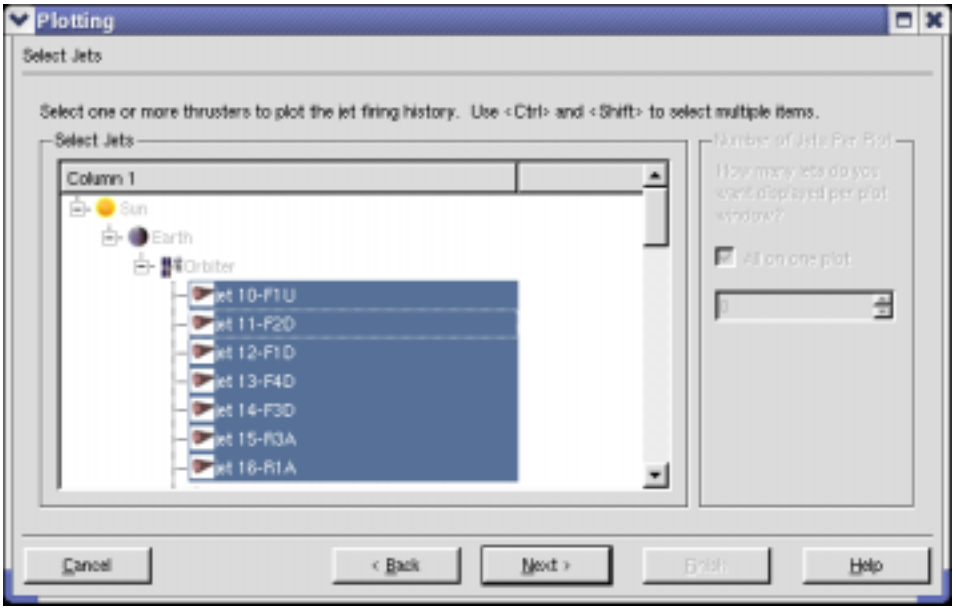

Figure 41. Selecting Jets for Plotting

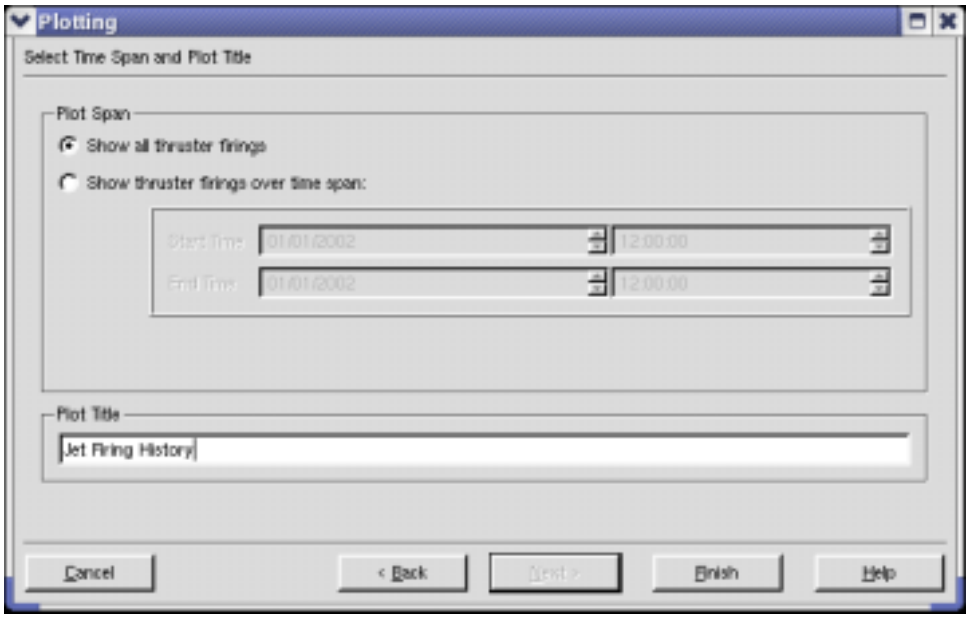

Figure 42. Specifying Plotting Times

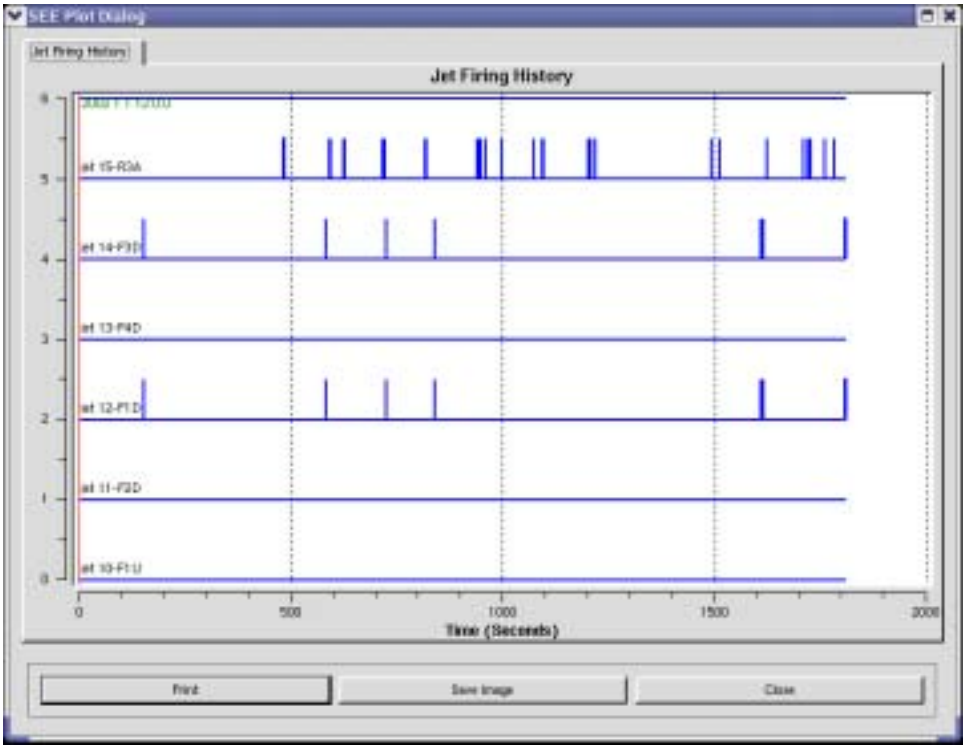

Figure 43. Jet Firing Plot

4.9 Range display

The SEE provides the ability to measure the distance between the center of any two objects in the scene. By selecting the Range display from the Tools menu, as was seen in Figure 29, the window shown in Figure 44, will be created. The user can select an item from the "From" menu and the "To" menu to see the distance between the objects. Various units can be selected from the units pulldown.

Figure 44. Range Display

4.10 Media Captures

Additional tools available to the user are the ability to capture an image or movie. At this time, movie capture is currently only supported under the Win32 release of the SEE. These capabilities can be found under the Media menu, as shown in Figure 45. Additionally, a snapshot quick access button is provided under the top toolbar.

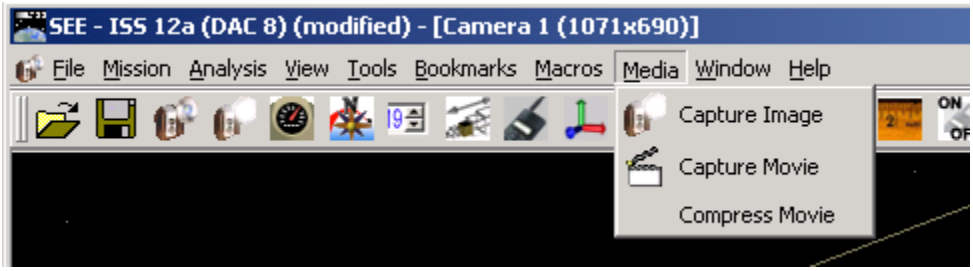

Figure 45. Media Menu

4.10.1 Capturing an Image

To capture an image, the user first sets up the view and window size. The window size will be used to determine the overall image size. Next the user selects either the toolbar icon or the Capture Image item from the Media Menu. This will capture the image and save it to the images directory found in the users working area. The file format can be set to any of the supported file formats found on the Media tab of the Preferences window found under the File menu, as shown in Figure 46. The user can also select a compression quality setting for the image captures for those formats that support compression. The higher the percentage value the user sets, the image will have greater image quality but larger file size.

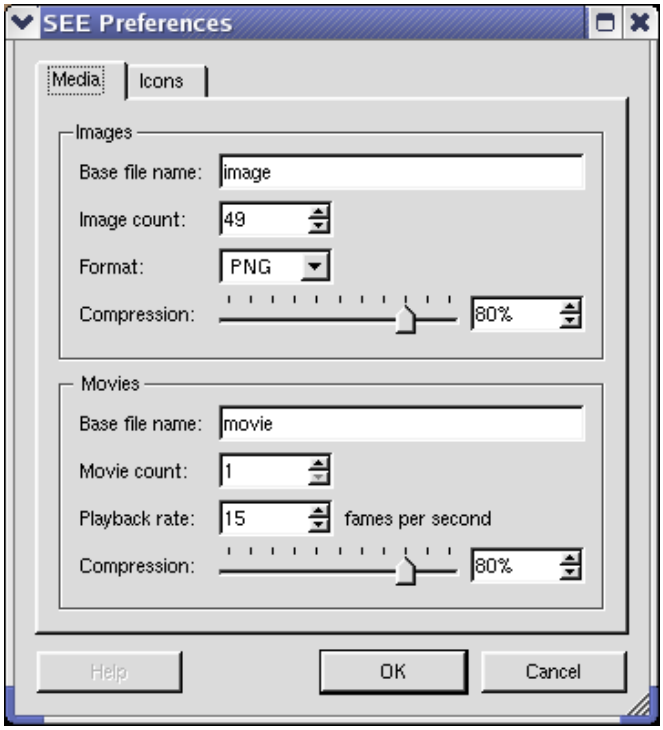

Figure 46. Image Type Preferences

4.10.2 Movie Capture

To create a movie, the user first sets up the scene to be captured. This includes setting up icons, setting a particular view, setting the time rates, and setting the window size. The user will have the ability to maneuver within the scene during the movie capture. The time rates set within the SEE will be preserved in the movie capture, although during the actual capture, the time rate may be different based on playback rate settings set in the Media tab of the preferences. This should be taken into account during any navigation performed during the movie capture. The window size will be used as a basis for the movie capture. This size combined with the compression quality setting can greatly affect the size of the movie file. Once the scene has been prepared, the user can select the Capture Movie item from the Media Menu, as was shown in Figure 45. This will bring up the window shown in Figure 47. The user must specify the start and ending times for the capture. The current time rates are displayed and can be modified. Additionally, the user has the option of compressing the movie during the capture of the images, or the user can capture without compression, and compress the movie in a post-processing mode. Once the user has set the parameters for the capture and presses the Record button, the SEE will take control of time and playback rate to create the movie at the desired rates. If the image was not compressed, the user will have the option of compressing the movie by selecting the Compress Movie option from the Media Menu and selecting the appropriating move file.

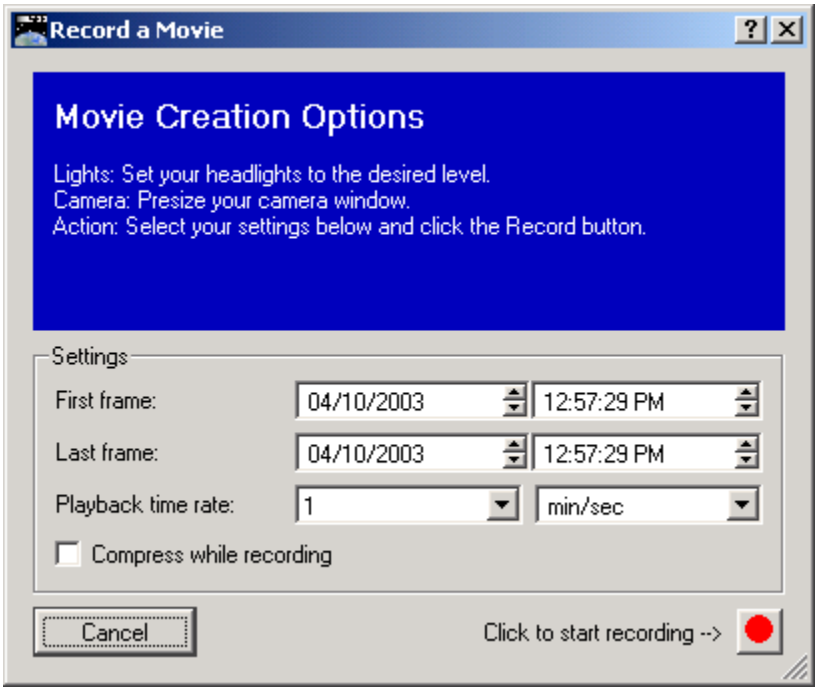

Figure 47. Movie Capture Settings

4.11 Display Properties

Various properties of the scene/camera can be changed by the user through the View Menu and sliders. The View Menu, shown in Figure 48, contains several toggleable menu items. First are the orbits of the planets, moons, and crafts. By pressing the orbits menu, and selecting one of the three choices, the orbit of that selected item will be either turned on or off. This can also be achieved by pressing the orbit button on the horizontal toolbar. This method walks through a series of states for the orbits: all orbits on, moons off, moons and planets off, all orbits off, all orbits on. Next is the ecliptic plane. By selecting this option a plane is drawn on the ecliptic. This option is useful for visualizing large-scale trajectories. The lines of sight utility can be toggled from the View menu. This will not remove the line from the Lines of Sight setup, only from being displayed at this time. Lastly, the stars can be toggled by selecting that option from the View Menu.

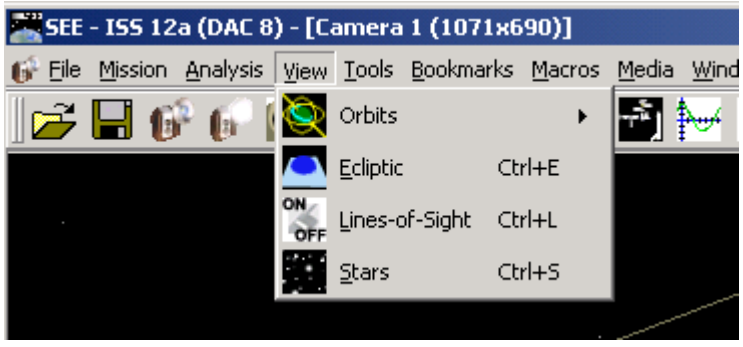

Figure 48. View Menu

The top right slider of the vertical menu, as was seen in Figure 14, can control both a headlight and the field of view of the camera. If the object being viewed is currently in shadow, an artificial headlight can be turned on from the users current position. The pulldown menu below the slider can be set to the headlight icon, and the slider can be adjusted for the amount of light required. If the pulldown is set to the magnifying glass, the slider is then used to adjust the field of view of the camera. This has the appearance of moving the camera but it is not. It is only adjusting the field of view. It is the equivalent to the user of a zoom lens.

5 Craft Manipulation

Once a craft has been loaded into the SEE application, the various properties that define the behavior of the craft and its orbit can be edited. This is done by selecting the appropriate craft from the Mission Menu as shown in Figure 49. This will bring up the spacecraft properties window, shown in Figure 50, allowing the user to edit the orbit, attitude, articulation behaviors, graphics, model, mass properties and assembly structure of the craft. The first tab allows the user to redefine the name of the craft, edit the primary body which the craft orbits, edit the reference frame the orbit data refers to, and add a description.

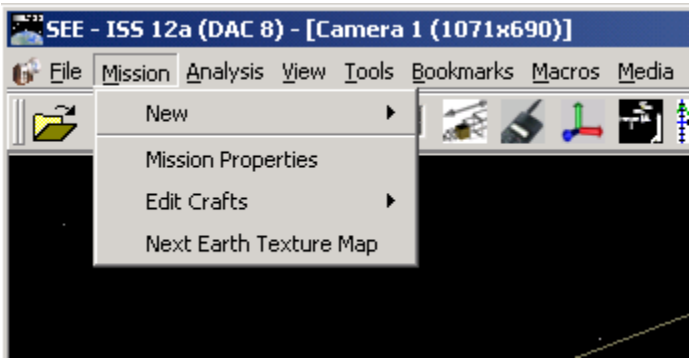

Figure 49. Edit Mission Menu

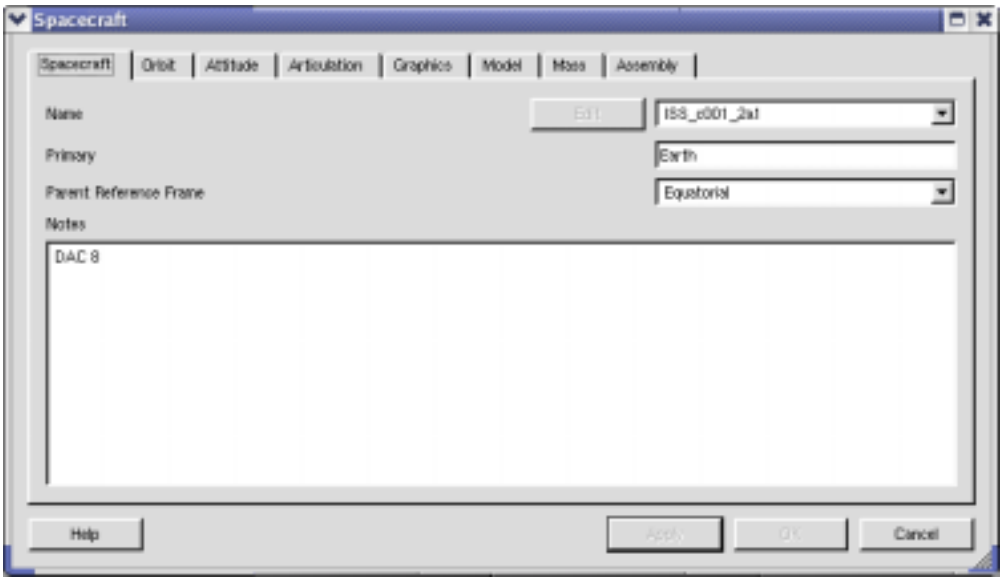

Figure 50. Edit Spacecraft Window

5.1 Orbits

The orbit parameters tab of the edit spacecraft window can be seen in Figure 51. This window provides the user the ability to define the orbit or trajectory of the spacecraft through several different means. This figure shows the use of a Kepler data source, where the user defines the six orbital elements. Discrete data can also be used to define the trajectory of the spacecraft. This requires the selection of a datafile that contains the location and orientation of the spacecraft in the given reference frame. For each type of data source, the user must also provide the Epoch time.

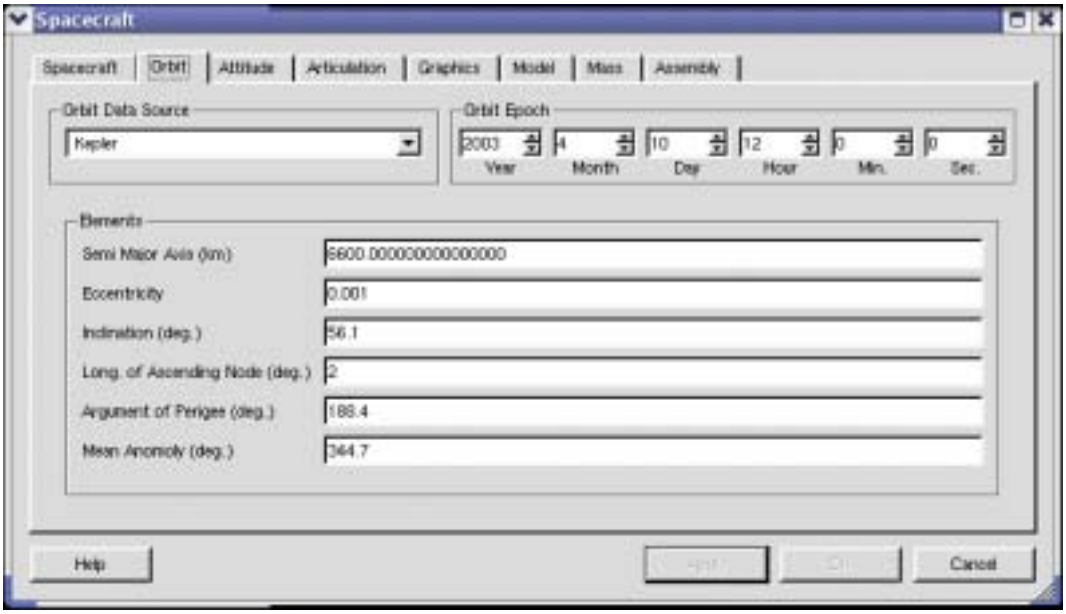

Figure 51. Orbit Properties

5.2 Attitude

The attitude tab of the spacecraft properties window can be seen in Figure 52. This window provides the user the ability to modify the attitude relative to the base orbit/trajectory data. The first property, the Flight Mode, allows the user to change the base orientation of the spacecraft. The XVV and XPOP flight modes are provided. The user can also specify an attitude offset using an Euler angle sequence. A name for the attitude file can be specified to store the specific setup within the mission file.

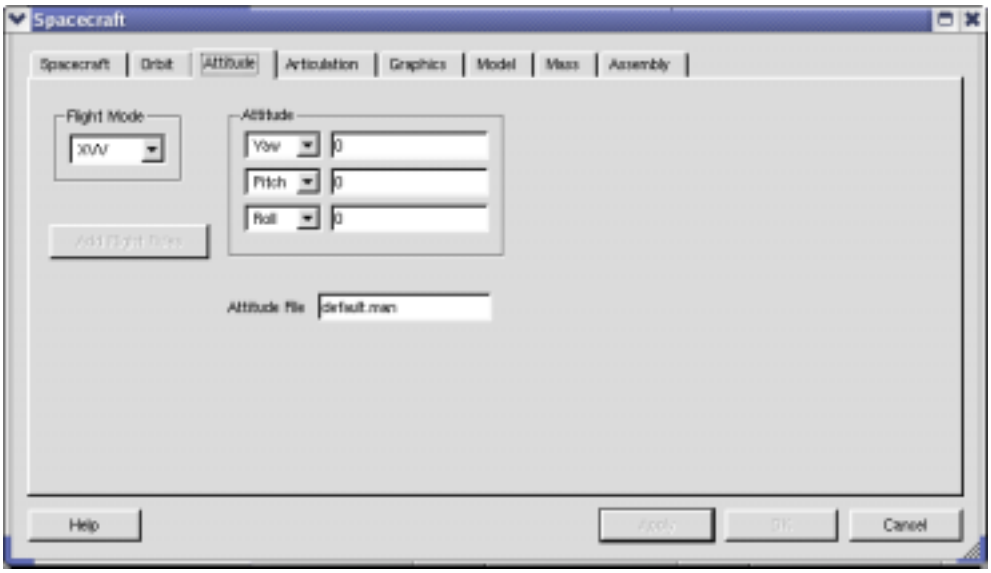

Figure 52. Attitude Properties

5.3 Articulation

The articulation tab of the spacecraft properties window provides the user the ability to specify the motion of any defined rigid bodies on a spacecraft. The main window of this tab can be seen in Figure 53. Any defined articulation will be shown on this main window. By pressing the Add Rules button, the user can add an articulation rule to the craft. The user will be presented the Articulation Wizard, as seen in Figure 54. The first step of the wizard is to select the rigid body or set of rigid bodies that the user want to apply a rule to. Once these have been selected and next is pressed, the user must select the mode of articulation. Currently, only absolute positioning is provided.

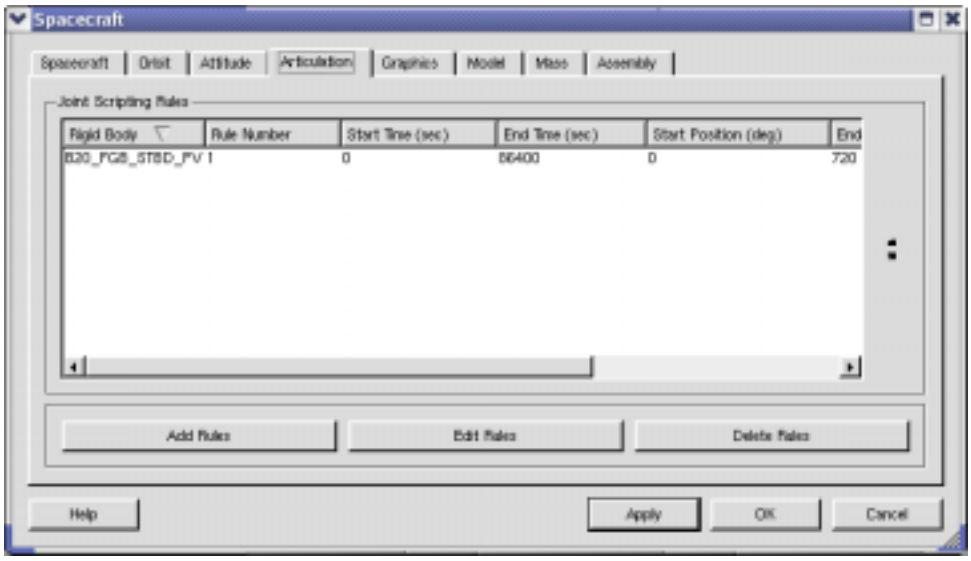

Figure 53. Articulation Properties

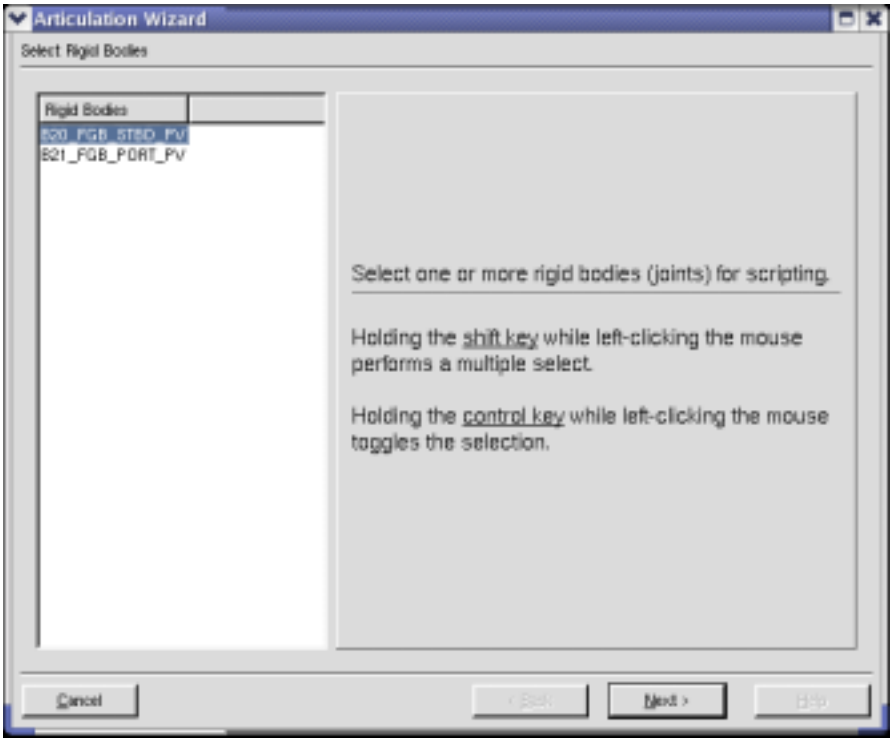

Figure 54. Articulation Wizard Selecting Rigid Bodies

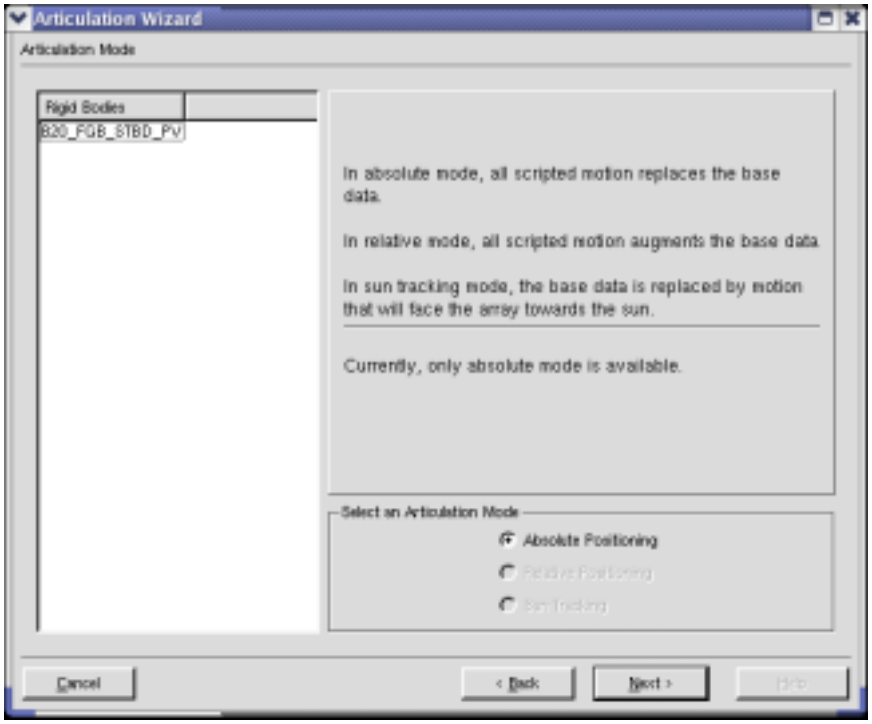

Figure 55. Articulation Wizard Selecting Mode

The next step in defining the articulation of the joints is the specification of a starting time for the motion. This can be seen in Figure 56. The user enters in a starting time that is relative to the craft's epoch time. For instance, a starting time of zero means that the joint motion starts at the same time the orbit/trajectory data begins. Next the user must enter the ending time. This is entered in the same fashion as the starting time. However, if the user does not know the ending time, but rather a rate of motion, the user can uncheck the box labeled "Ending time known". Last, the user must enter a starting position and ending position as seen in Figure 58 and Figure 59. Once the finish button has been selected, the rule will be added to the list in the articulation tab.

To remove a rule or set of rules, the user can select them on the articulation tab window and press the delete rule button. To edit a rule or set of rules, the user can select the rule(s) from the articulation tab window and press the edit rule button. This will step the user through the same process as creating the rule. Only rules that were created at the same time can be edited at the same time.

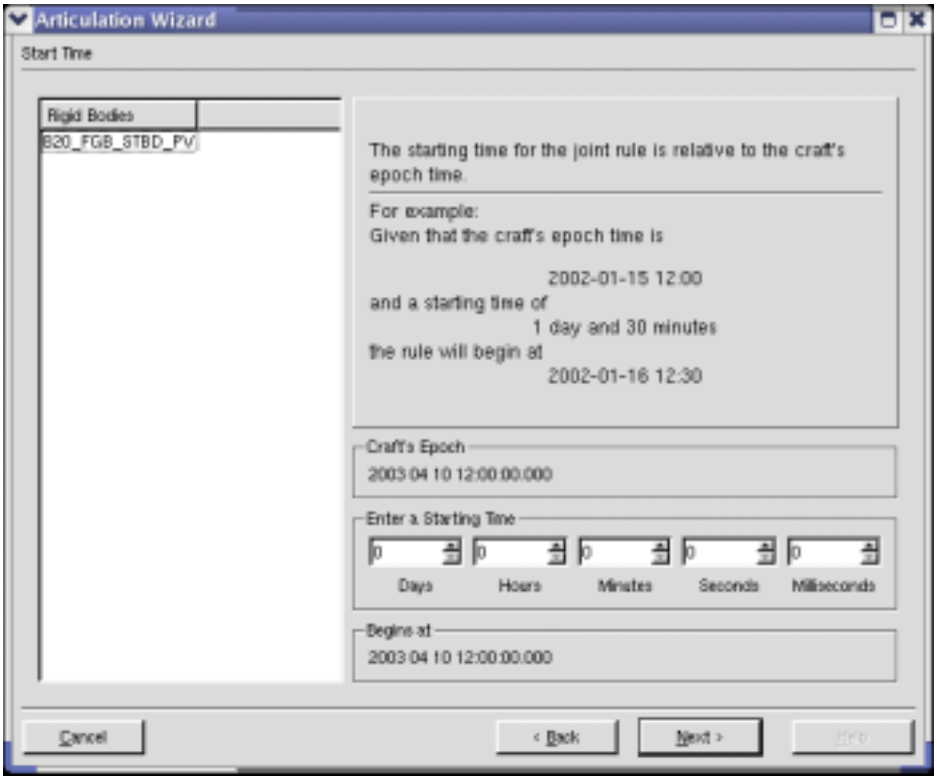

Figure 56. Articulation Wizard Starting Time

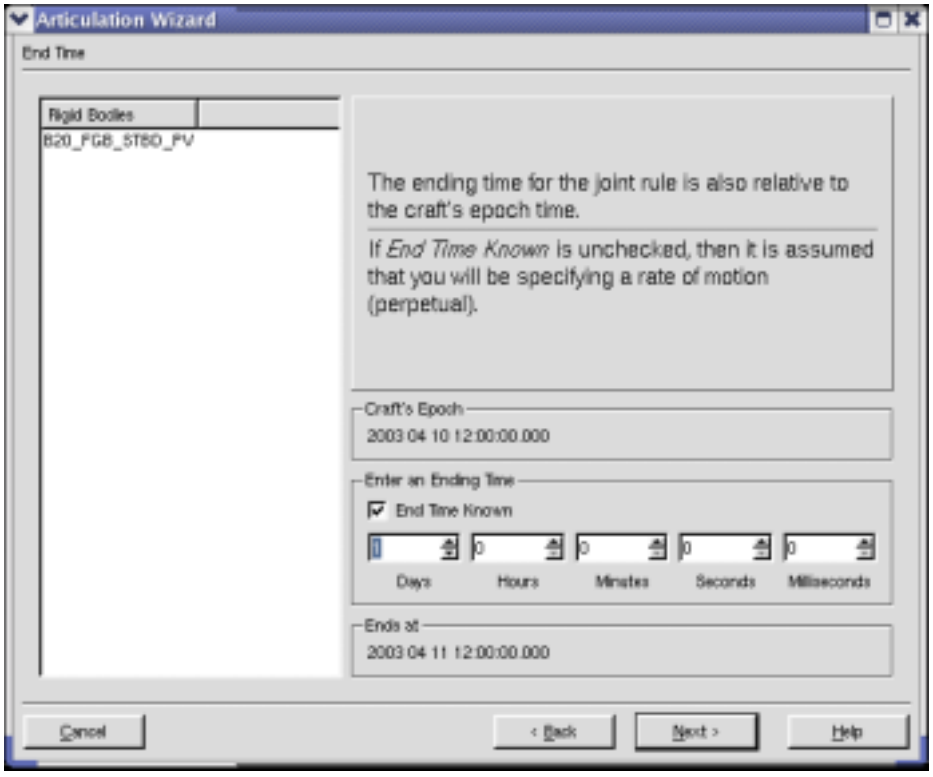

Figure 57. Articulation Wizard Ending Time

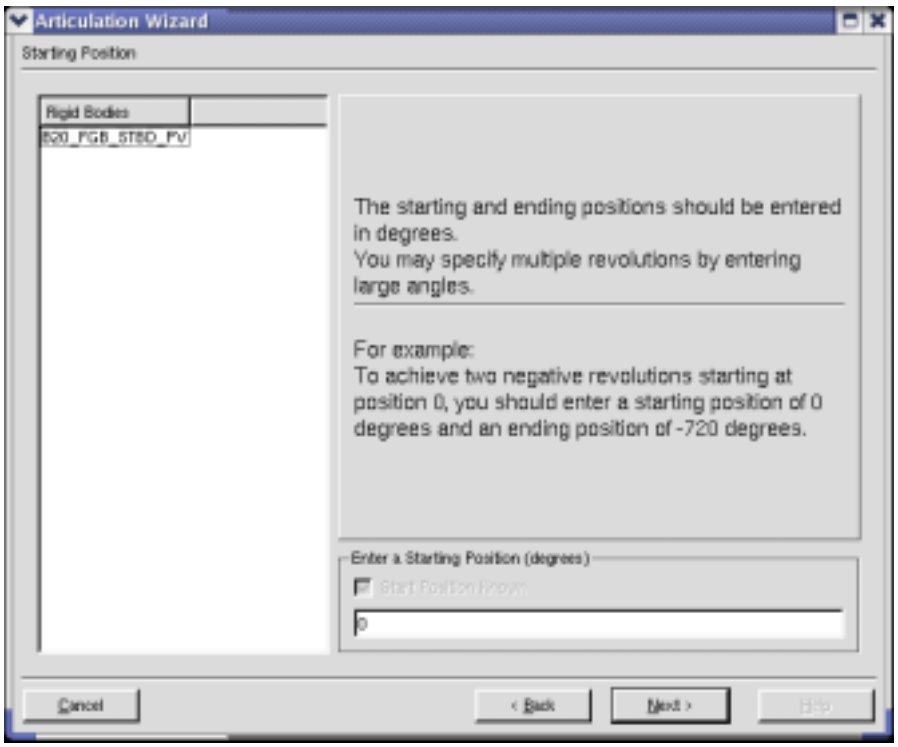

Figure 58. Articulation Wizard Starting Position

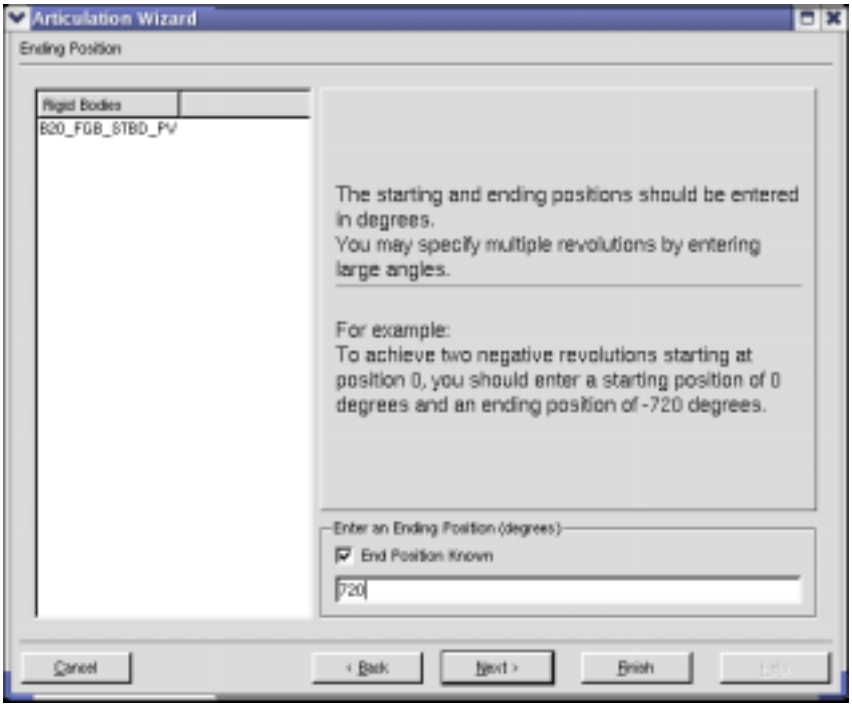

Figure 59. Articulation Wizard Ending Position

5.4 Graphics

On the graphics tab of the spacecraft properties window, the user can edit the color of the orbit as it will be drawn in the scene. Additional properties will be modifiable in future releases. This window can be seen in Figure 60.

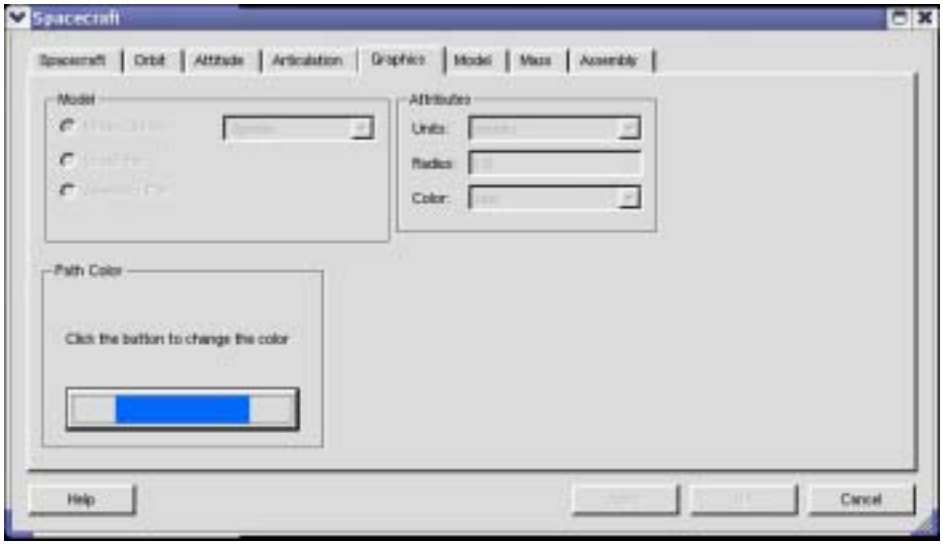

Figure 60. Graphics Properties

5.5 Model

The model tab of the spacecraft properties window is used to modify the position and orientation of the various parts and assemblies of the spacecraft. This window can be seen in Figure 61. On the left side of the window is a hierarchical display of the spacecraft assemblies and parts. By selecting an assembly or part from the list, the current values for position and orientation are shown on the right. To locate a part, the user can press the highlight button. The specific assembly or part will be shown in red in the scene until the highlight button is unselected. To move the selected assembly or part, the user can enter in values in either the translate or rotate sections of the window. By pressing the test button, the user can see the change made to the model without committing the change.

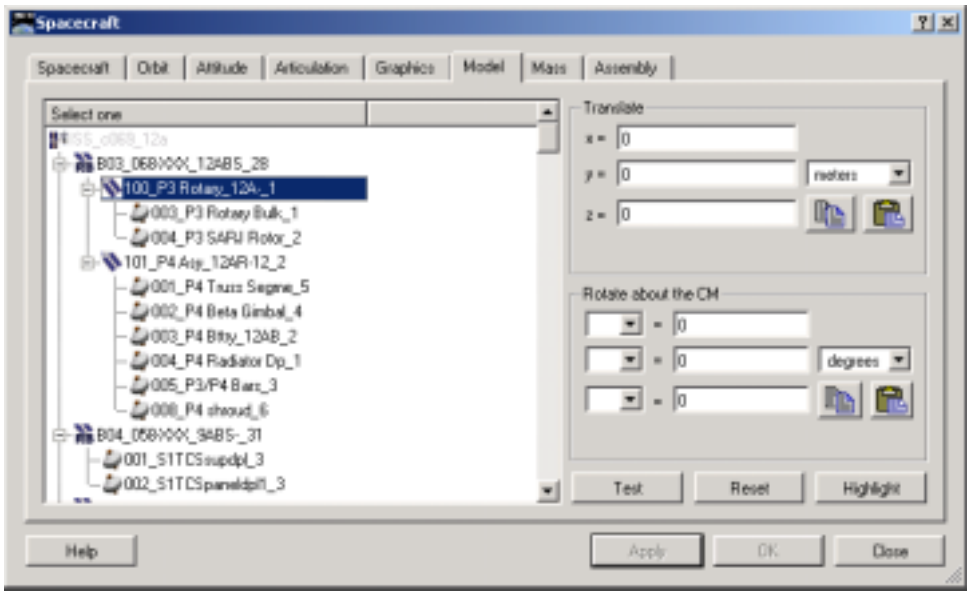

Figure 61. Model Properties

5.6 Mass

Each spacecraft can have mass properties associated with it. The mass tab of the spacecraft properties window shown in Figure 62, allows the user to see these values. The hierarchical structure of the assemblies and parts is shown in the left portion of the window. By selecting on an assembly or part, the current values of the mass properties are shown. These include the mass, center of mass, and inertia properties.

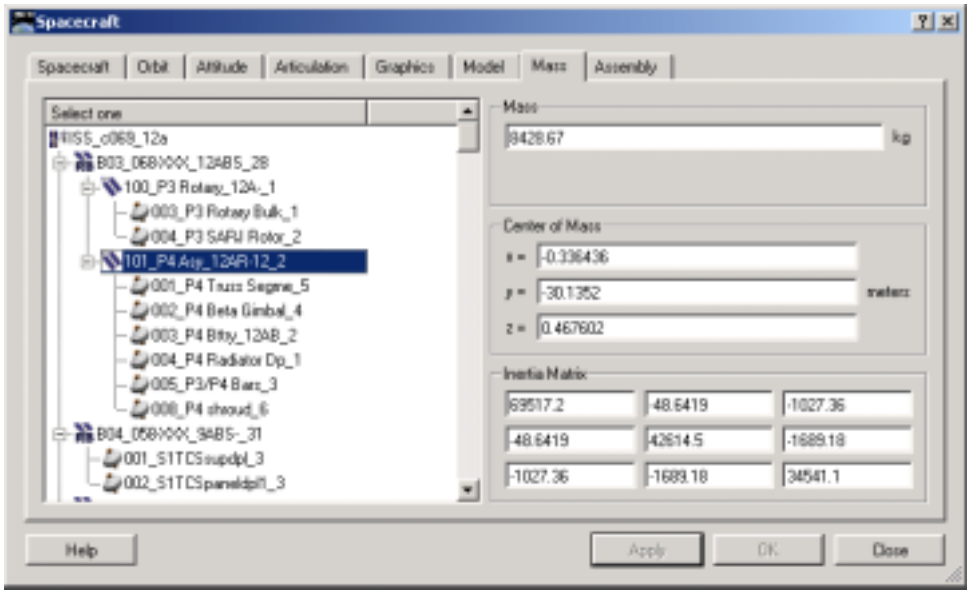

Figure 62. Mass Properties

5.7 Assembly

The last tab of the spacecraft properties window is the assembly tab, as shown in Figure 63. This window allows the user to add, remove, or re-parent assemblies and parts. Figure 63 shows the Add part capability. The user must first select an assembly to add the part to. Then the user must provide a name and model file to be loaded. The units for the model must be specified to ensure a consistent size within the environment.

To remove a part the user must select Remove on the window as well as the part or assembly to be removed. This window can be seen in Figure 64. Currently the ability to re-parent a part or assembly is not supported by the SEE.

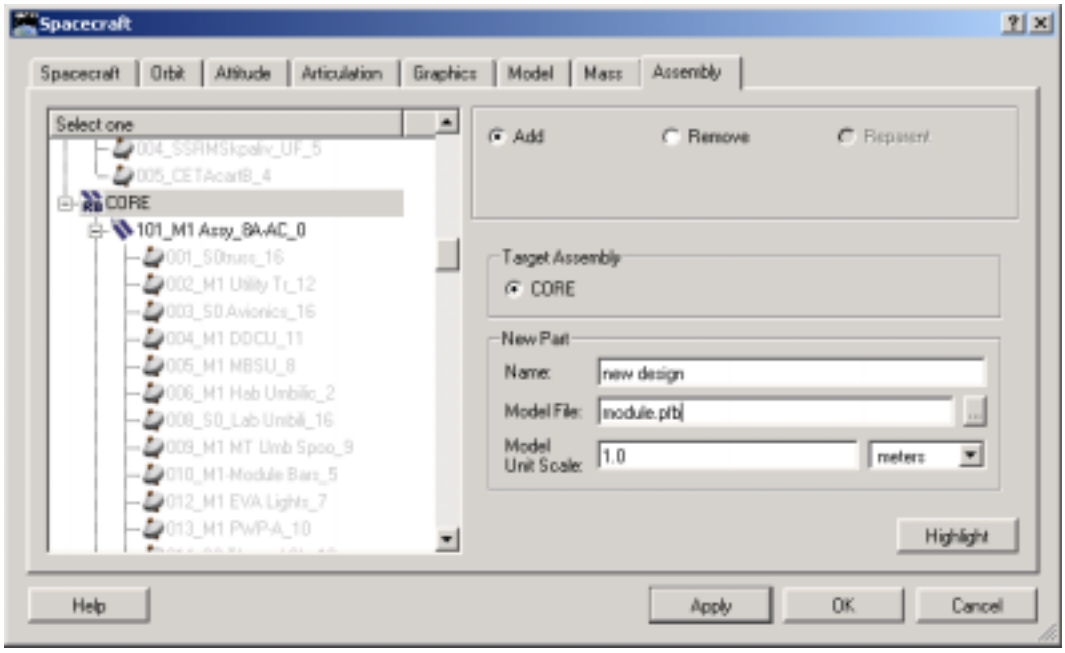

Figure 63. Assembly Properties

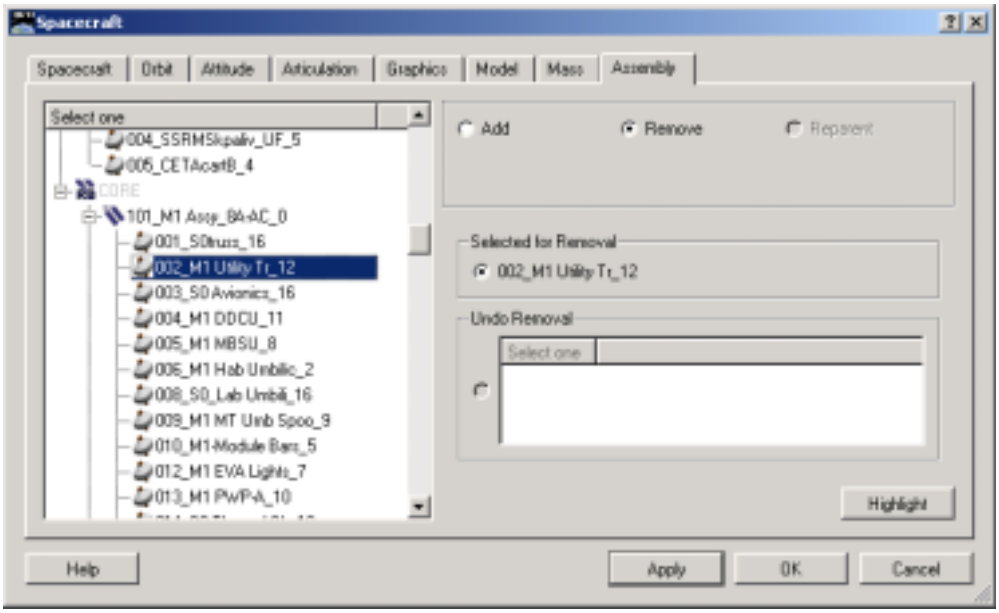

Figure 64. Removing Assemblies and Parts

6 Advanced Capabilities

6.1 Integrated Analyses

Several analysis capabilities have been built into the SEE. These include ARCD for rigid body dynamics analysis, collision detection and proximity analysis tool, a point to point line of sight analysis tool, and a suite of tools for creating and analyzing a comet and asteroid protection system. These tools can be found under the Analysis menu as shown in Figure 65.

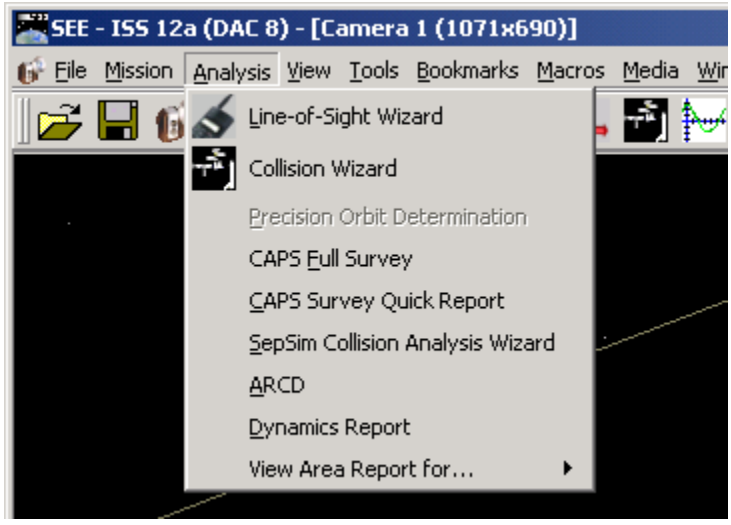

Figure 65. Analysis Menu

6.1.1 ARCD

ARCD is a tool that has been developed to analyze the torque equilibrium attitude and momentum build up a spacecraft. To use ARCD, the user must select the ARCD option from the Analysis menu and select a craft from the list of objects in the scene, as shown in Figure 66. Once this is done, the ARCD analysis tool window, shown in Figure 67 will be displayed. The first tab, spacecraft, provides the current setup of the spacecraft that was selected from the list. This includes orbit and epoch information. The rigid body tab provides the user the ability to setup each rigid body set of parameters. These parameters include attitude and rotation information. Each rigid body must be set independently by selecting on the rigid body on the left side of the window and modifying its parameters on the right side of the window. Once all parameters are set, the user can select the finish button. This will run the ARCD software with the specified data. The output will be read back into the application for review and analysis.

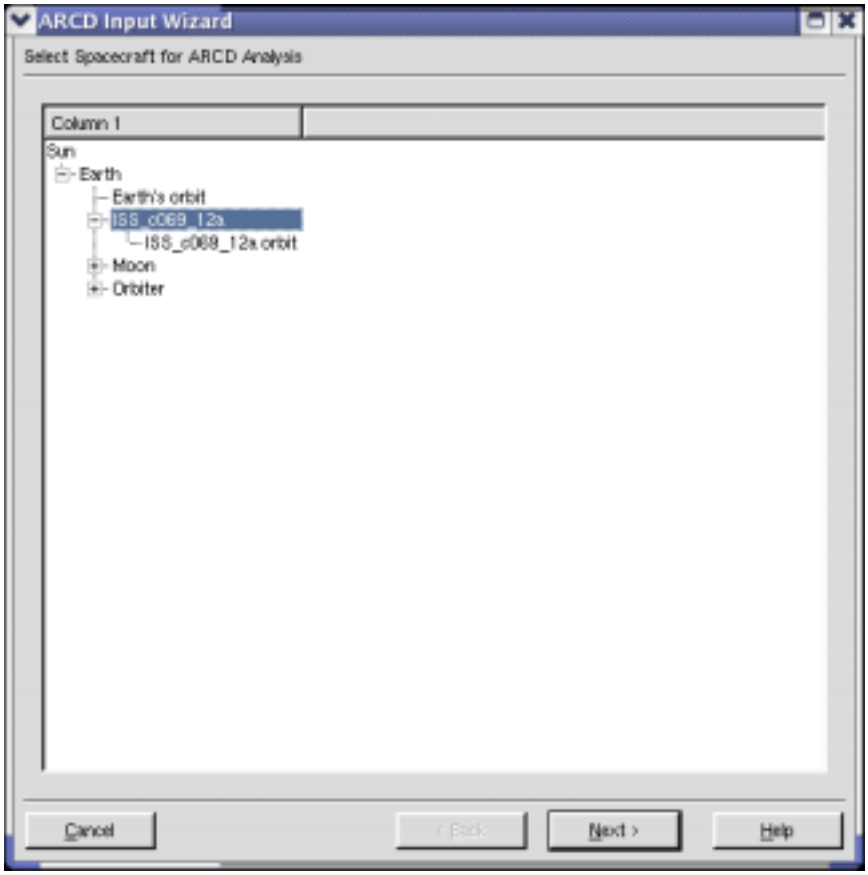

Figure 66. ARCD Wizard Selecting Craft

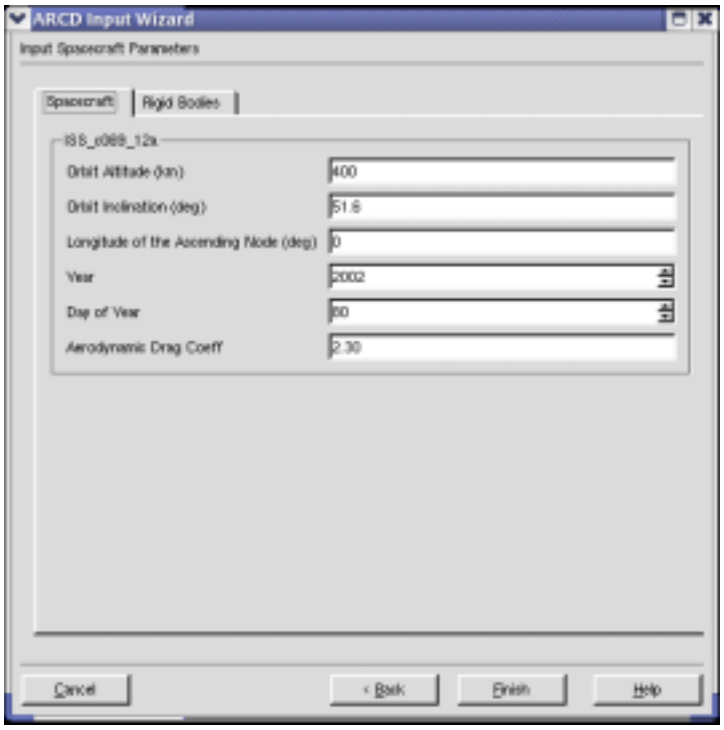

Figure 67. ARCD Analysis Window

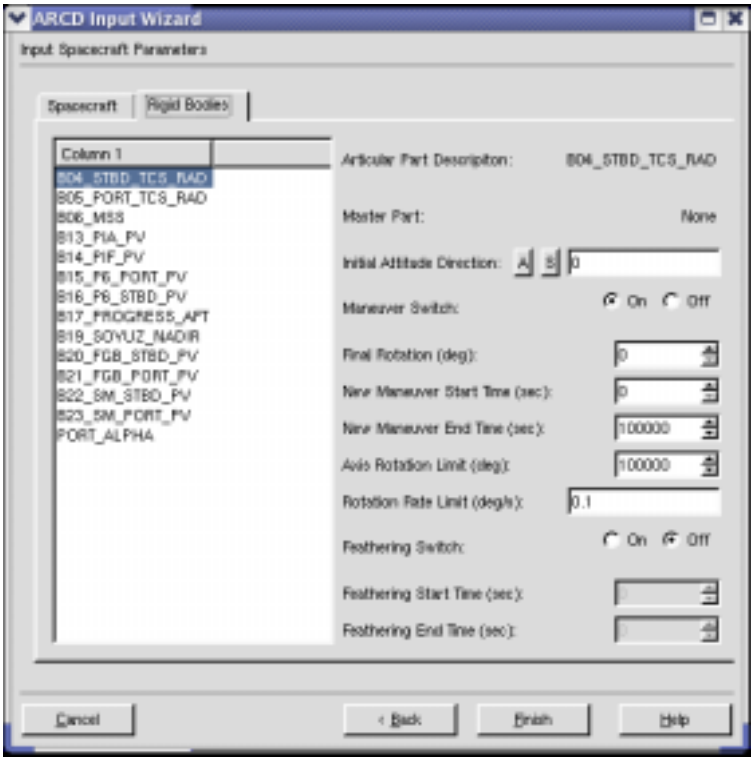

Figure 68. ARCD Rigid Body Window

6.1.2 Collision Detection

Integrated into the SEE is the ability to perform various proximity analyses on the various spacecraft loaded into the application. These include collision testing, tolerance testing, and proximity testing. A collision test only notifies the user if and when a collision occurs between two sets of objects defined by the user. A tolerance test notifies the user if and when the two sets of objects get within a user-defined distance from one another. A proximity test constantly monitors the closest point between two sets of objects. The user has the option of running these analyses either in a real-time interactive mode or an analysis mode. In a real-time mode, the simulation will run at the rate specified by the user on the time interface. The specific frame rate will determine the simulation time interval at which the collision tests are performed. In an analysis mode, the user enters in the time steps at which the collision tests are performed. This guarantees that a test is done for each specified step. An output file is created for review if an analysis is performed in this manner.

By selecting the collision wizard from the analysis menu, the user is presented the window shown in Figure 69. By selecting any of the tests to be run in interactive mode and pressing next, the user must define the object sets for the analysis. This can be seen in Figure 70. Here the user can select any number of rigid bodies, assemblies, or parts from the list on the left portion of the window. Then by selecting one of the right-pointing arrows, can add them to either the first or second collision groups. If an object is incorrectly added, it can be reselected on the right side and removed by selecting the left-pointing arrow.

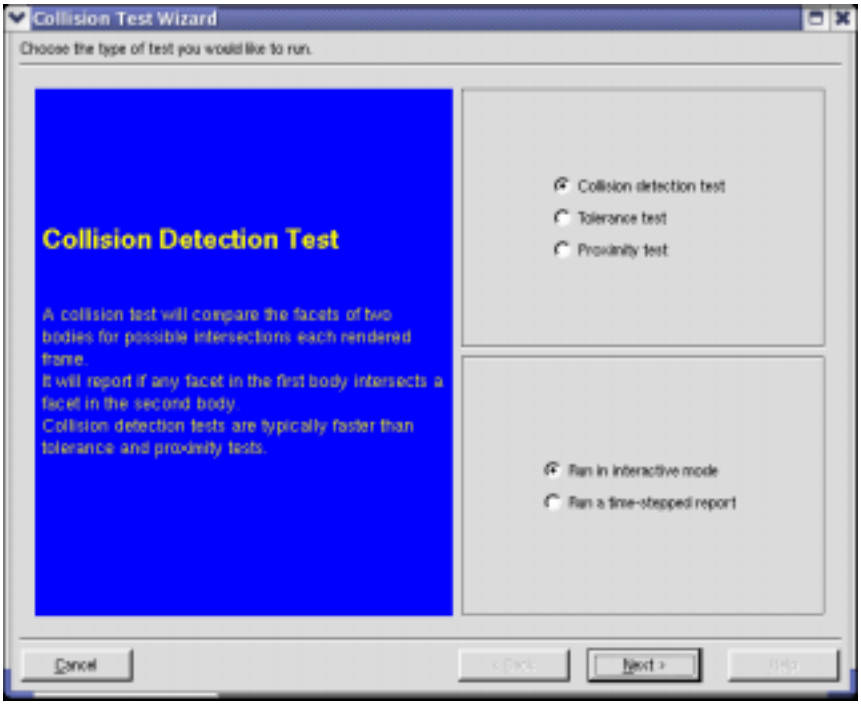

Figure 69. Collision Wizard

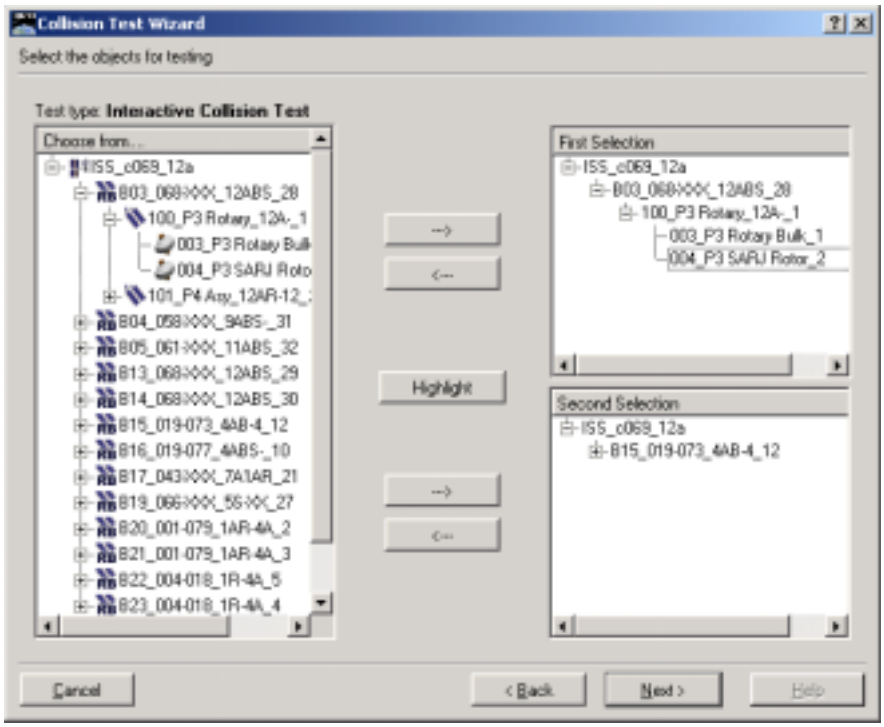

Figure 70. Collision Wizard Object Selection

If the user had selected a time-based analysis instead of interactive, the user must first enter in the starting time, ending time, and time step for the analysis. This can be seen in Figure 71. Then the user would select the object sets for the analysis as described above.

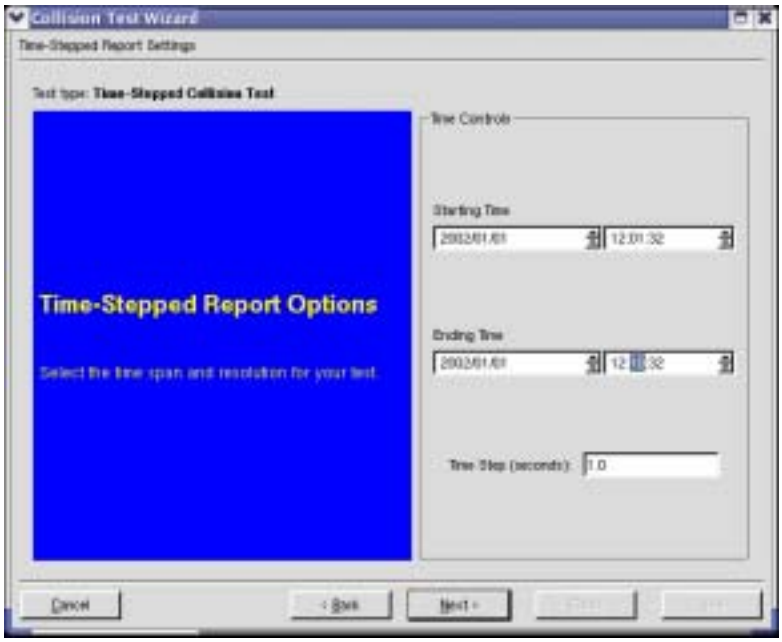

Figure 71. Collision Wizard Time-Stepped Report

Next, if the user had selected either the tolerance or proximity options, they must set a tolerance value as shown in Figure 72. Additionally, the user must set some preferences for feedback to the user during the tests. This includes a visual line between the closest points, a display of the current closest distances, and the colors for greater than and less than the tolerance distance.

The next step is for the user to setup the behavior of the application in case a collision or tolerance violation has occurred. These options include sound notification, popup window notification, and simulation time behavior. These options can be seen in Figure 73.

Once the analysis has been setup and finish has been selected, the application prepares the necessary data for the analysis. This may take a few moments. Once the preparation is done, the application window will return and a collision/proximity toolbar is added to the toolbar section of the main interface, as shown in Figure 74. This toolbar allows the user to pause the analysis, shows the status of the analysis (color indications for collision or tolerance violations), and allows the user to cancel the test. By clicking on the arrow on toolbar, the toolbar will expand, such as that shown in Figure 75 for a collision analysis. This provides the user the ability to change some of the notification parameters. Tooltips will pop up as the user puts the cursor over each of the checkboxes that indicate which parameter is being modified.

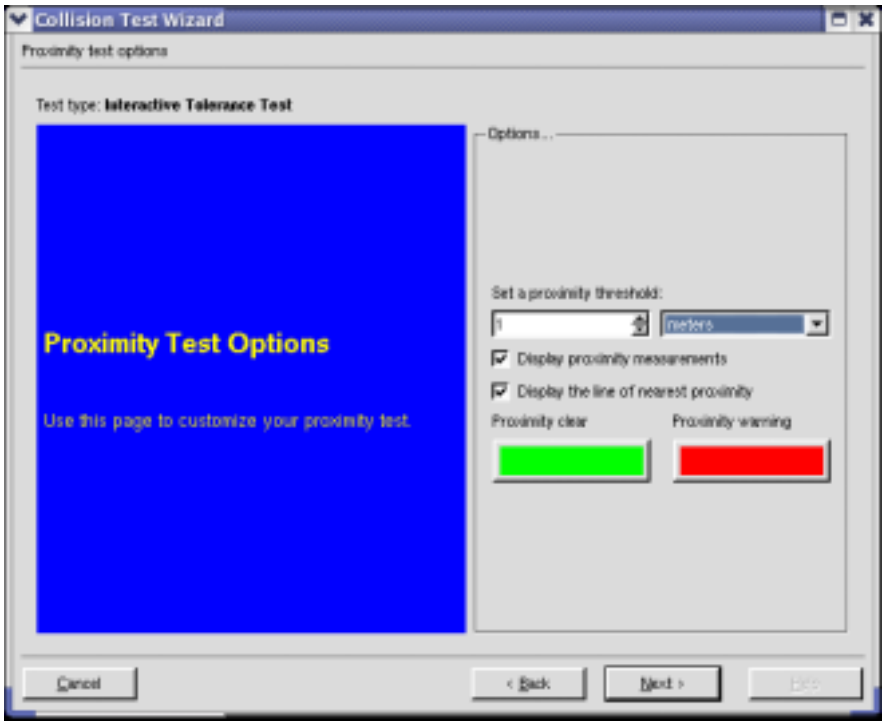

Figure 72. Collision Wizard Proximity Settings

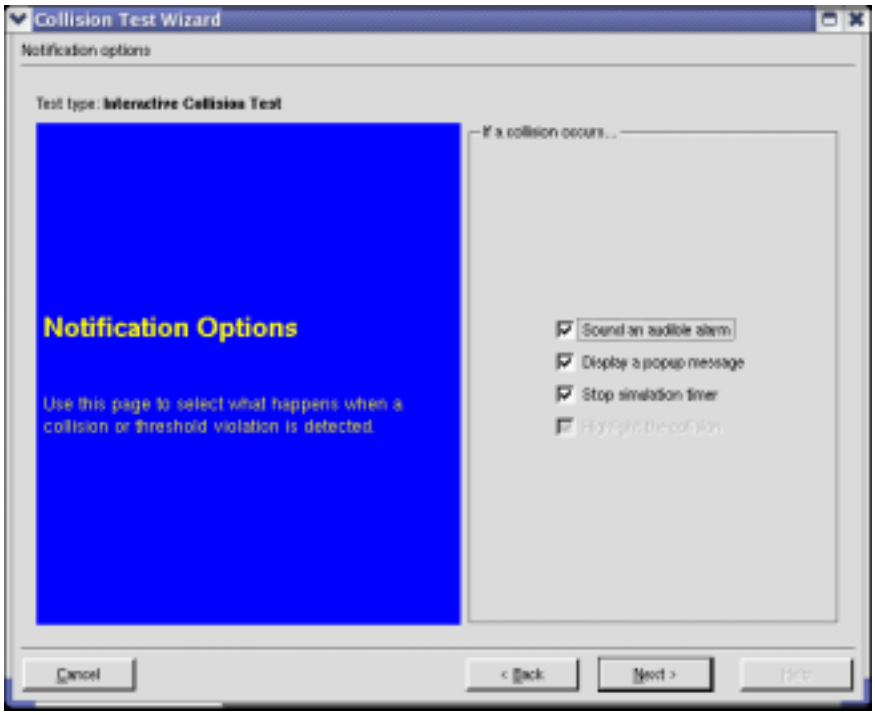

Figure 73. Collision Wizard Notification Options

Figure 74. Collision Analysis Status Toolbar

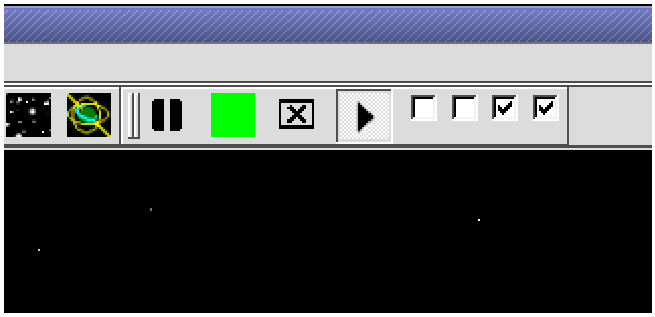

Figure 75. Collision Analysis Toolbar Expanded

6.1.3 Line of Sight Analysis

It is often desired to calculate or visualize the line of sight between different objects within the scene. This includes communications line of sight or solar rays for thermal concerns. This ability was added to the SEE to allow for such an analysis. The SEE allows the user to add one or more lines of sight between two objects, or multiple lines of sight from one object to many other objects. This section steps through the procedures and capabilities of the Line of Sight Analysis.

6.1.3.1 Load Mission

The first step in performing a line of sight analysis is loading in the proper mission. The mission should include all spacecraft, planets, and moons that are required for the analysis. All data pertaining to the trajectory of the spacecraft and motions of the spacecraft rigid bodies should be set for the time of interest for the line of sight analysis.

6.1.3.2 Start Line of Sight Wizard

To setup a line or set of lines for analysis, a wizard has been created to step the user through the process. This wizard can be launched by selecting the Line of Sight Wizard from the Analysis pulldown menu, as seen in Figure 76.

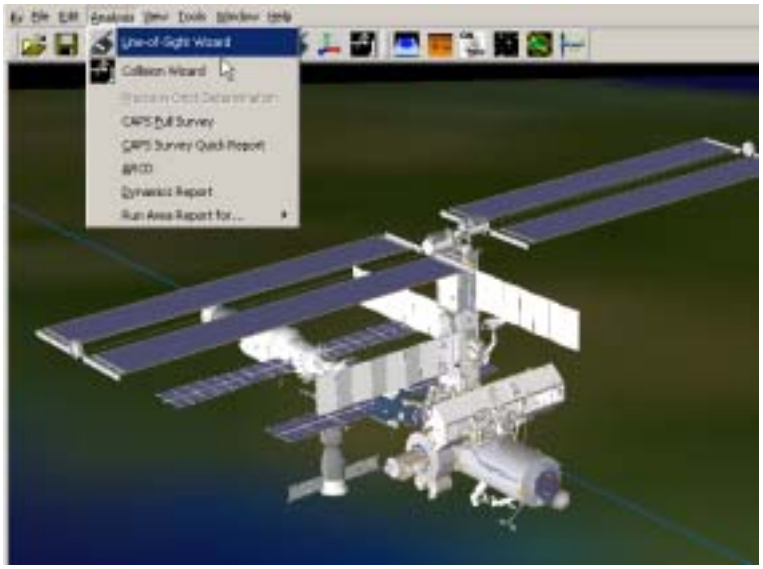

Figure 76. Analysis Pulldown Menu

6.1.3.3 Adding Line(s)

Once the wizard has been launched, the user is presented with the main window, as seen in Figure 77. Four selections are available at this point: Add Lines, Edit Lines, Remove Lines, and Run Analysis. By selecting the Add

Lines option, the user is presented with the next window, asking for the origin object. This can be seen in Figure 78. The object can be any planet, moon, craft, rigid body, assembly, or part within the scene. After selecting the origin, the user must select one or more targets. This window can be seen in Figure 79. By selecting one target, the user will have the option to have multiple lines between the origin and target and have offsets for each end of line at the origin and target. If the user elects to have multiple targets, then only offsets will be available at the origin for all lines.

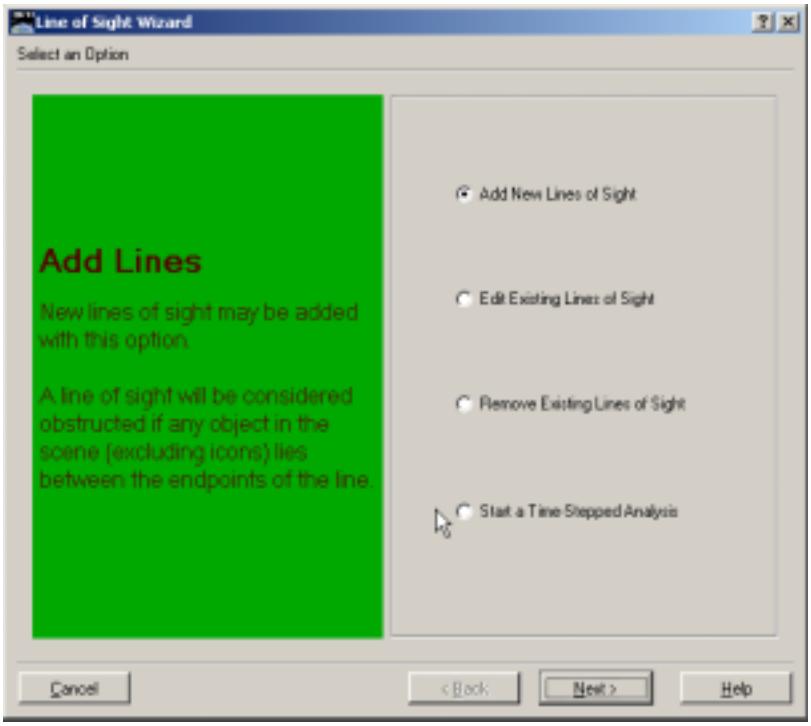

Figure 77. Main Line of Sight Window

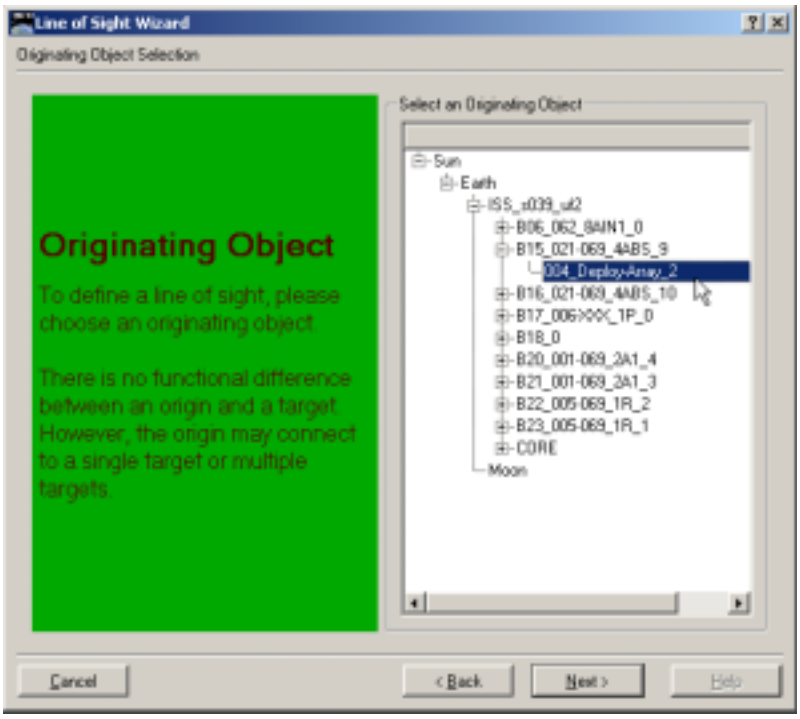

Figure 78. Selecting Originating Object

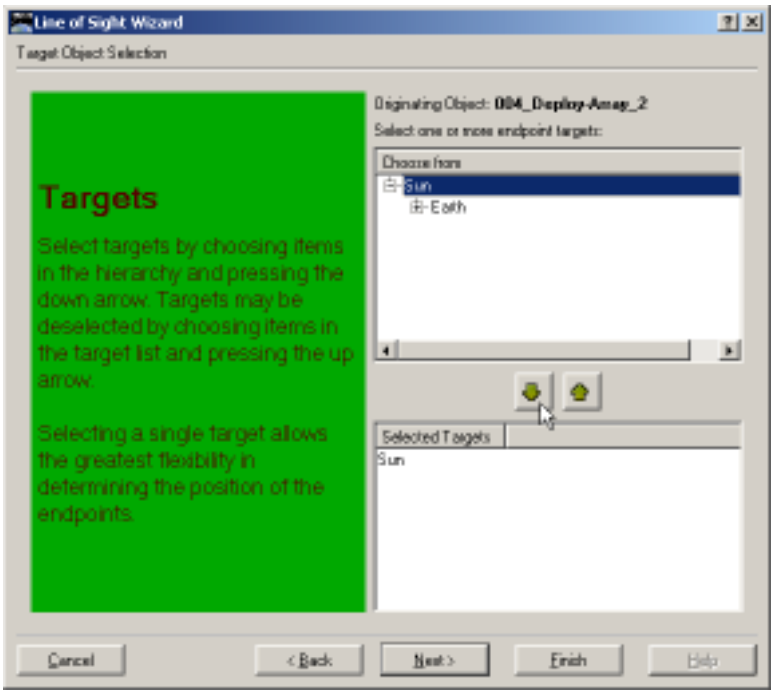

Figure 79. Selecting Target Object(s)

6.1.3.4 Setting Offsets

After selecting the originating and target objects, the user has the option of adding offsets to the endpoints of the lines. The offset window is shown in Figure 80. If only one line is being added to the scene, the user directly enters in the offsets on the offsets window. The user must first select which frame of reference within the object the offset should be applied. All available offsets can be viewed and selected using the pulldown list. By default the synchronous reference is used. The next step is to enter in the offset for the endpoints. The user can enter XYZ rectangular coordinates or latitude, longitude, and altitude information. All latitude, longitude, and altitude data is converted to XYZ once entered. The altitude data is relative to the bounding sphere of an object and is most applicable to adding points of interest to planets and moons. An additional setting that can be set on this window is the exclusion of either the originating object or target object from the line of sight test. For instance, a line of sight from a spacecraft to the sun will most likely want the craft to be included in the test, but exclude the sun itself.

The last option the user has on this page is the ability to load multiple lines that will go between the originating and target objects. This is done by selecting a text file using the browse capability. This file is a simple space delimited text file that specifies an originating offset and a target offset for each line. These offsets are placed relative to the reference frame chosen within the window for each object. A sample file format can be seen in Figure 81. The first three numbers in a line are an offset for the originating endpoint and the last three are an offset for the target object. Any lines starting with a '#' sign are ignored.

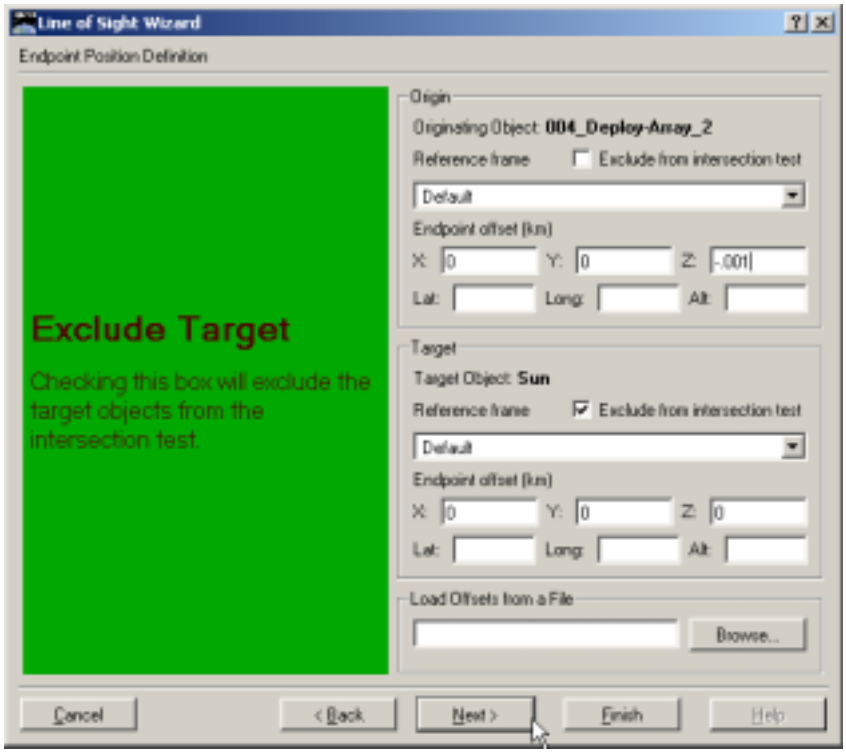

Figure 80. Adding Offsets

#	x	y	z	x	y	z
	.001	0	0	.002	0	.003
	0	.001	0	.002	0	.004
	0	.002	0	.002	0	.004

Figure 81. Sample Text Offset File

6.1.3.5 Setting Colors

At this point the user can select the Finish button and the lines will be displayed within the scene using the default colors for clear versus blocked line of sight. Or if the user wishes, by pressing the Next button, the user can change the default colors, as shown in Figure 82. This is done by selecting the line from the list of lines, and then selecting the color chooser for either the clear or blocked color. After selecting the color and selecting Finish, the line(s) will be displayed.

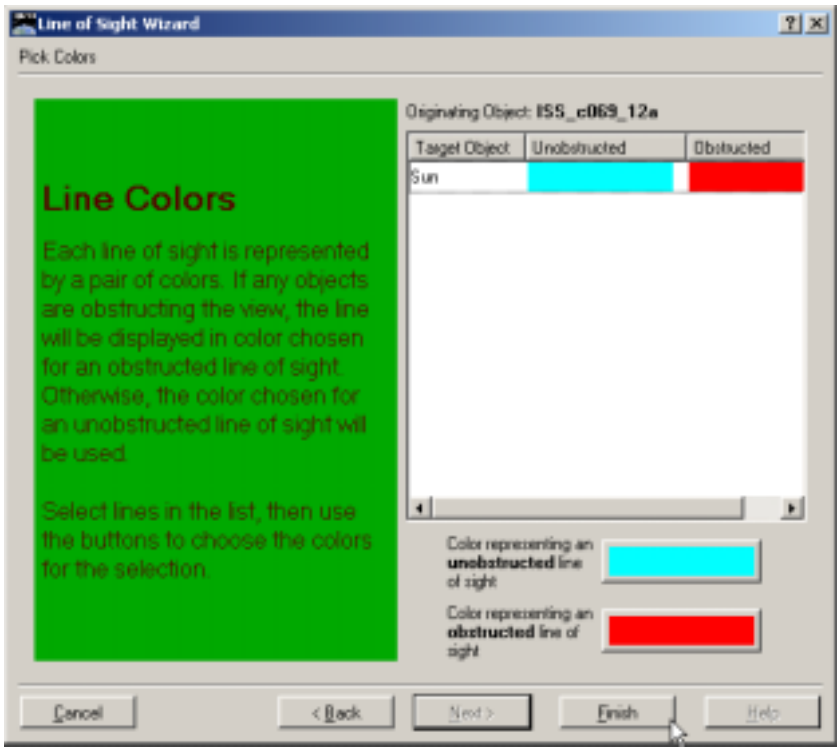

Figure 82. Setting Colors

6.1.3.6 Editing Lines

Once all lines are in the scene, the user can change the properties of any of these lines. By selecting the Line of Sight Wizard, the user will be presented with the main wizard screen. By selecting the second choice, the user will be presented with the window shown in Figure 83 to select a line to change. Only one line can be changed at a time and all of the attributes can be changed. After making the appropriate changes and pressing the Finish button, the lines will be updated in the scene.

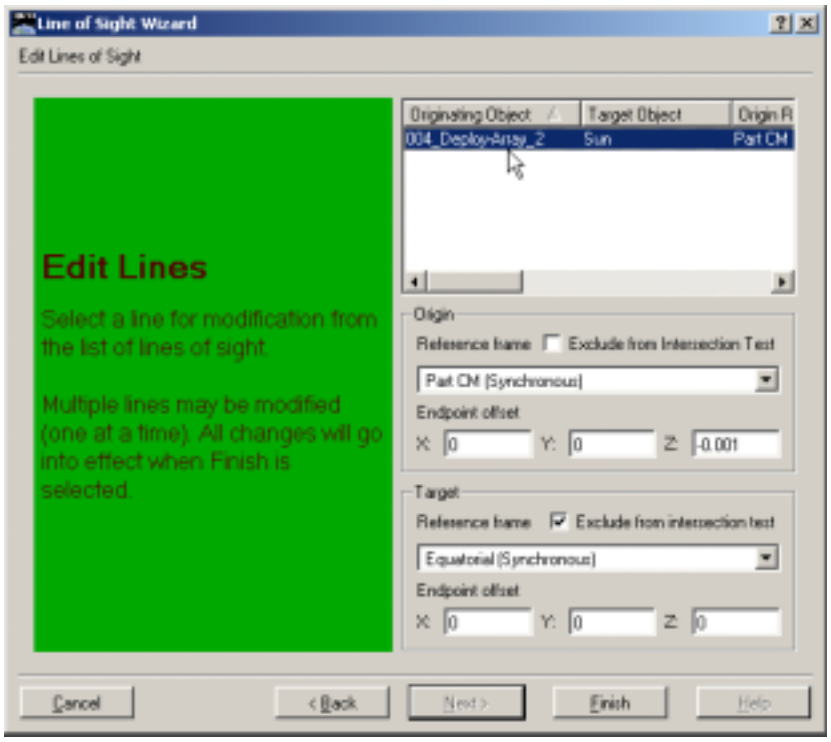

Figure 83. Editing Lines

6.1.3.7 Removing Lines

To remove a Line of Sight, the user selects the Line of Sight Wizard to get the main window. On this screen, selecting the third choice will present to the user the window shown in Figure 84 to select lines to delete. One or more lines can be selected for deletion. This will remove the lines from the scene and all future analyses.

6.1.3.8 Running Analysis

Once the user has all of the desired lines of sight into the scene, an analysis can be performed. By selecting the fourth option from the main Line of Sight Wizard, the user will be presented with an analysis setup window, as seen in Figure 85. This window lets the user set the starting time, ending time, and time step for calculating the line of sight for each line that the user currently has setup within the SEE. The current time in the application is the default starting time. Once the time settings have been entered and the user presses the Finish button, the application will automatically set the time to the start time, run the application for each time step until the end time is reached, and then display the results. For each line of sight in the scene, a horizontal bar will be created that shows the state of the line of sight at every time step. Additionally, these results are output to two data files. These are the setup/summary and the results files.

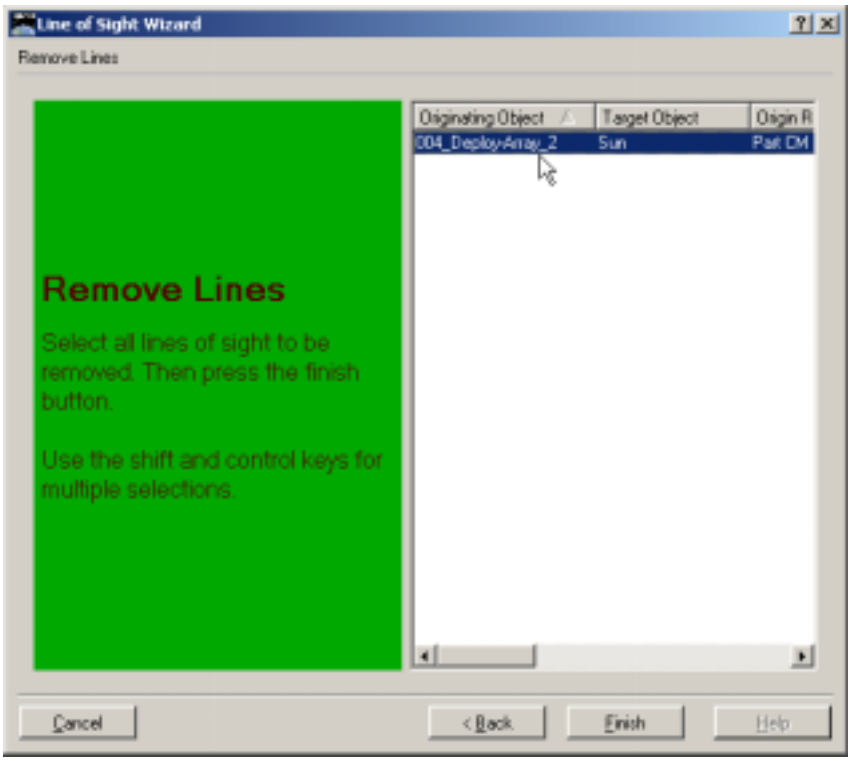

Figure 84. Removing Lines

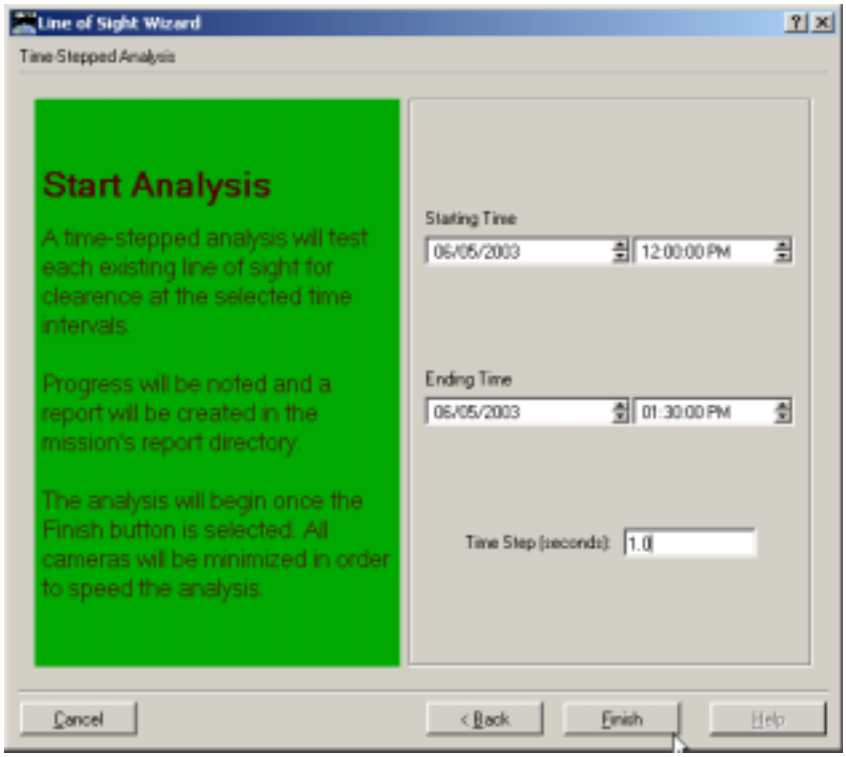

Figure 85. Analysis Window

The summary file contains the parameter setup for the analysis and each of the lines within the scene. This includes the associated objects, reference frames, offsets, and representative row in the results file. A sample setup file can be seen in Figure 86. The results file is a space delimited column-based file. The first column is a line count. The second column is the time step, starting with time zero for the first row and incrementing each row by the time step for the analysis. The third column is the absolute time for the time step in Julian format. After the Julian time is a series of columns, one for each line in the scene. For the given time step, a zero or one is entered. A one represents a clear line of sight and zero represents a blocked line of sight. A sample can be seen in Figure 87.

```
[Time Stepped Line of Sight Analysis]
Start_Time (y-m-d h:m:s) = 2003-04-10 12:29:25.000
End_Time (y-m-d h:m:s) = 2003-04-10 12:34:25.000
Time_Step (seconds) = 10
Counter_Column = 1
Relative_Time_Column (seconds) = 2
Julian_Date_Column (days) = 3

[Line Of Sight]
Column = 4
Origin = 014_POF PV Dpl_1
Origin_Reference_Frame = Part CM (Synchronous)
Origin_Offset (km) = ( -0.011, -0.003, -0.0001 )
Origin_Excluded = No
Target = Sun
Target_Reference_Frame = Equatorial (Synchronous)
Target_Offset (km) = ( 0, 0, 0 )
Target_Excluded = Yes
[Line Of Sight]
Column = 5
Origin = 014_POF PV Dpl_1
Origin_Reference_Frame = Part CM (Synchronous)
Origin_Offset (km) = ( -0.001, -0.003, -0.0001 )
Origin_Excluded = No
Target = Sun
Target_Reference_Frame = Equatorial (Synchronous)
Target_Offset (km) = ( 0, 0, 0 )
Target_Excluded = Yes
```

Figure 86. Sample Setup and Summary File

```
    0         0    2452740.0204282408  1  1
    1        10    2452740.0205439813  1  1
    2        20    2452740.0206597224  1  0
    3        30     2452740.020775463  1  0
    4        40    2452740.0208912035  1  0
    5        50    2452740.0210069446  0  0
    6        60    2452740.0211226852  0  0
    7        70    2452740.0212384257  0  0
    8        80    2452740.0213541668  0  0
    9        90    2452740.0214699074  1  1
   10       100    2452740.0215856479  1  1
   11       110     2452740.021701389  1  1
   12       120    2452740.0218171296  1  1
   13       130    2452740.0219328706  1  1
   14       140    2452740.0220486112  1  1
   15       150    2452740.0221643518  1  1
   16       160    2452740.0222800928  1  1
   17       170    2452740.0223958334  0  1
   18       180     2452740.022511574  0  0
   19       190     2452740.022627315  0  0
   20       200    2452740.0227430556  0  0
```

Figure 87. Sample Results File

6.1.4 CAPS

Integrated into the SEE is an integrated analysis capability for the Comet and Asteroid Protection System (CAPS) Project of the Revolutionary Aerospace Systems Concepts Activity. Please refer to the CAPS Module Users Guide, AMA Report Number 02-30, for a detailed explanation of this capability.

6.2 Plume Cone Visualization

The SEE has the ability to display the plumes of jet firings. This is displayed as a cone with the apex of the cone located at the jet location, and the base of the cone draw in the direction of the jet firing. The width and length of the cone represent the extents of the plume at time of firing. Jet firings are set up for each craft and mission using a series of data files. The details on setting up this data can be found in the SEE Developer's Guide.

If a mission has been designed with jet firing data, the user can enable the visualization of these firings by selecting the Thruster Visualization item from the Tools Menu. This will bring up the thruster window as seen in Figure 88. Here the user has several options in visualization the thrusters. The transparency setting is used to control the opacity of the cone during the times when the thruster is not firing. This can be used to get a better sense of where the thrusters are located, as seen in Figure 89. The display settings enable the user to turn off the firing visualization without disabling the visualization of the location, or to use automatic or manual

sampling. The sampling mode controls how the SEE displays the firing of the cone during any playback rate set within the SEE. Therefore, the SEE will always show firings if the playback speed is much faster than the jet firing pulse rates. If a manual mode is chosen, the user controls the time "window", where this value in seconds tells the SEE to show the firings of any jets that are firing plus or minus this amount of time relative to the current time. The user has the option of changing this interface into a toolbar version by selecting the arrow at the bottom of the window. This can be seen in Figure 90. Selecting the arrow on the toolbar will re-display the window.

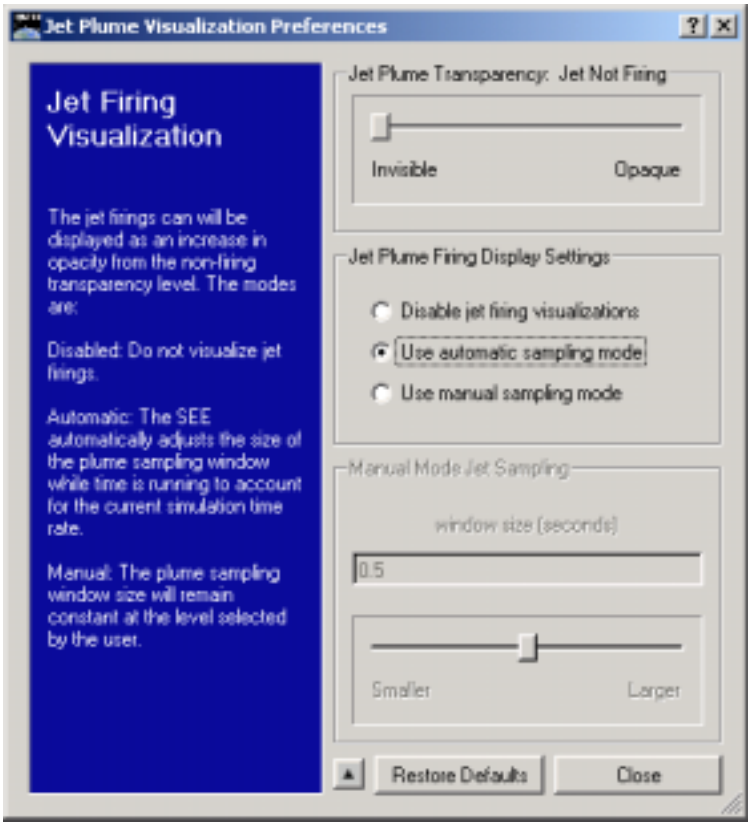

Figure 88. Thruster Plume Window

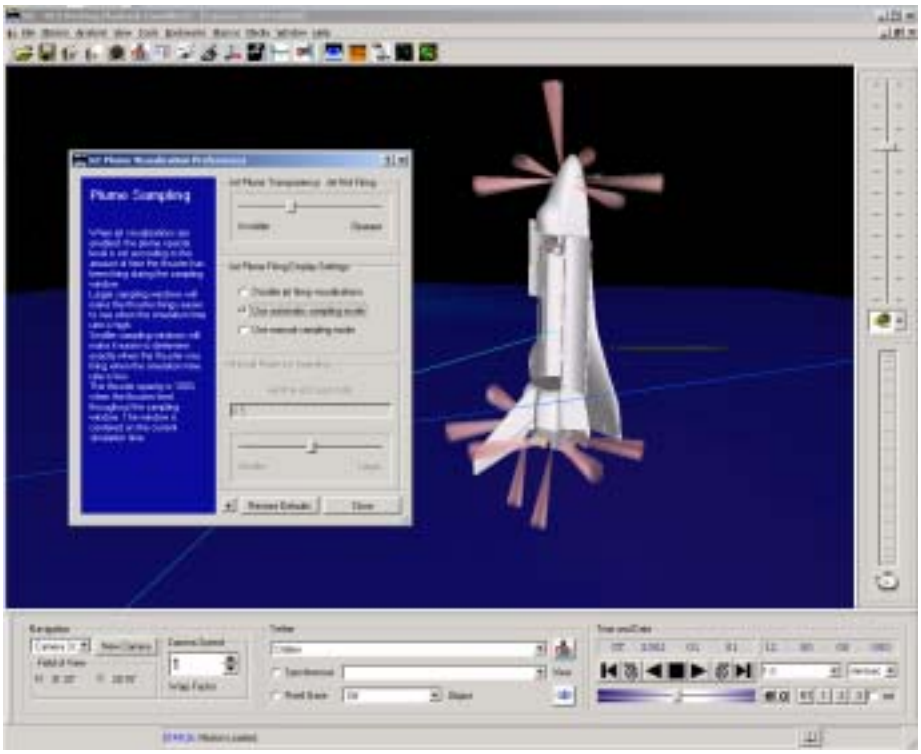

Figure 89. Visualizing all Thrusters

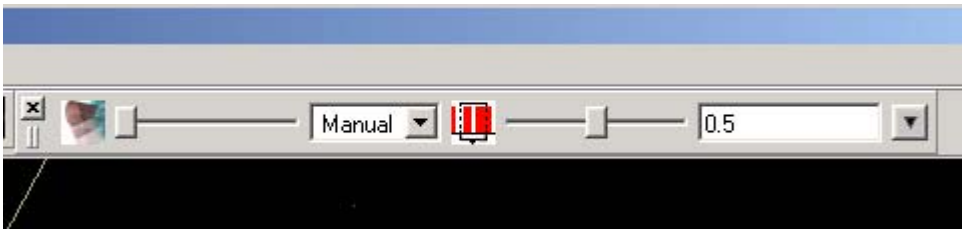

Figure 90. Thruster Visualization Toolbar

6.3 Macros

To facilitate the need to perform the same set of operations over and over within the SEE, a macro capability has been developed. This macro capability allows the user to store off a series of functions for future recall. Currently the SEE supports commands for loading data, setting up collision and proximity tests, copying files at the system level, changing several display properties, adding and removing icons, and creating line of sight tests.

To add a macro, the user selects the Add Macro item from the Macro menu as shown in Figure 91. This will display the main macro window as shown in Figure 92. The

user can name the macro, then proceed to add commands to be performed. To add a command, the user selects the Add button. This displays the window shown in Figure 93. On this window, the user selects the category of the command in the top pulldown list. Then a command is chosen from the second pulldown list. Any user required information will be prompted for in the lower portion of the window. When the user then selects the Add button, the command is added to the list of the main macro window. The user can re-order the items and delete commands as needed before accepting the macro. Once the macro is created, it is added to the Macro Menu, such as the "load data" macro shown in Figure 91.

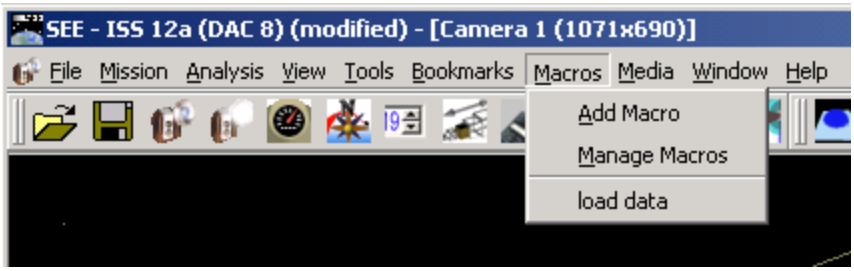

Figure 91. Macro Menu

Figure 92. Macro Window

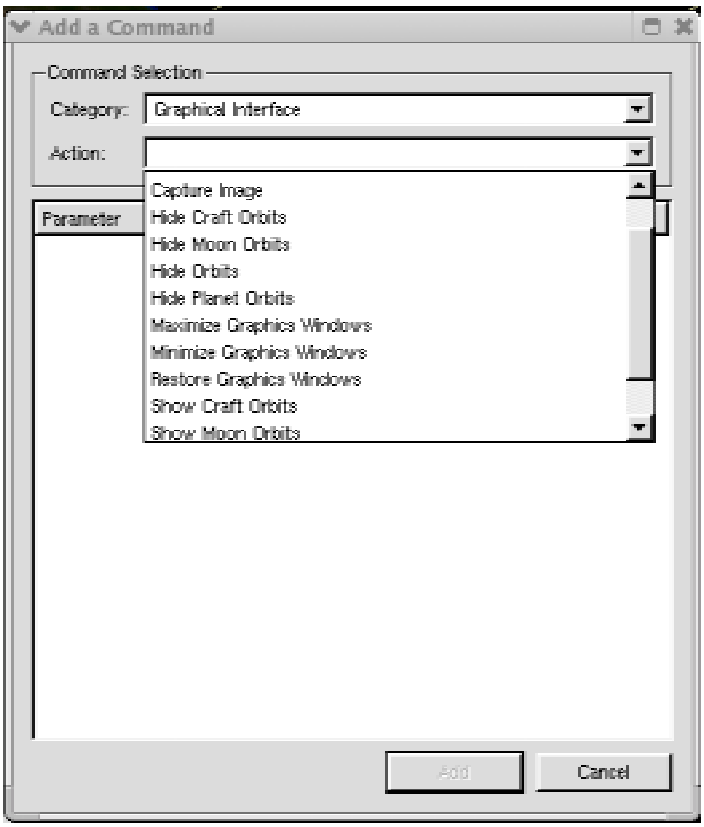

Figure 93. Add Macro Command